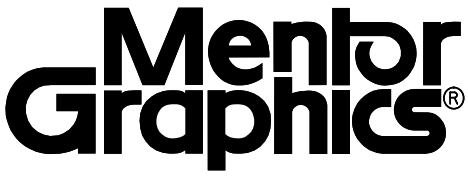
Board Station for New Users Training Series

Module 5: Placing Components on a Circuit Board

Software Version 8.5_2



Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

TABLE OF CONTENTS

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

Lesson 4

TABLE OF CONTENTS [Continued]

Lab 4

TABLE OF CONTENTS [Continued]

LIST OF FIGURES

LIST OF FIGURES [Continued]

LIST OF TABLES

About This Training

Welcome to the *Board Station for New Users Training Series*. For information on the tools you learn to use in this training series, see the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Workbook Organization

For an overview of the organization and content of all the modules of the *Board Station for New Users Training Series*, refer to section "Workshop Overview" in *Module 1: Introduction to Board Station.*

Related Documentation

For a complete listing of the manuals that make up the PCB documentation set, refer to section "Guide to the Documentation" in the *PCB Products Overview Manual*. The *PCB Products Overview Manual* describes how each manual can help you in the design process. You can find a listing of all Mentor Graphics manuals in the *Mentor Graphics Technical Publications Overview Manual*. Both of these manuals are available in INFORM.

Documentation Conventions

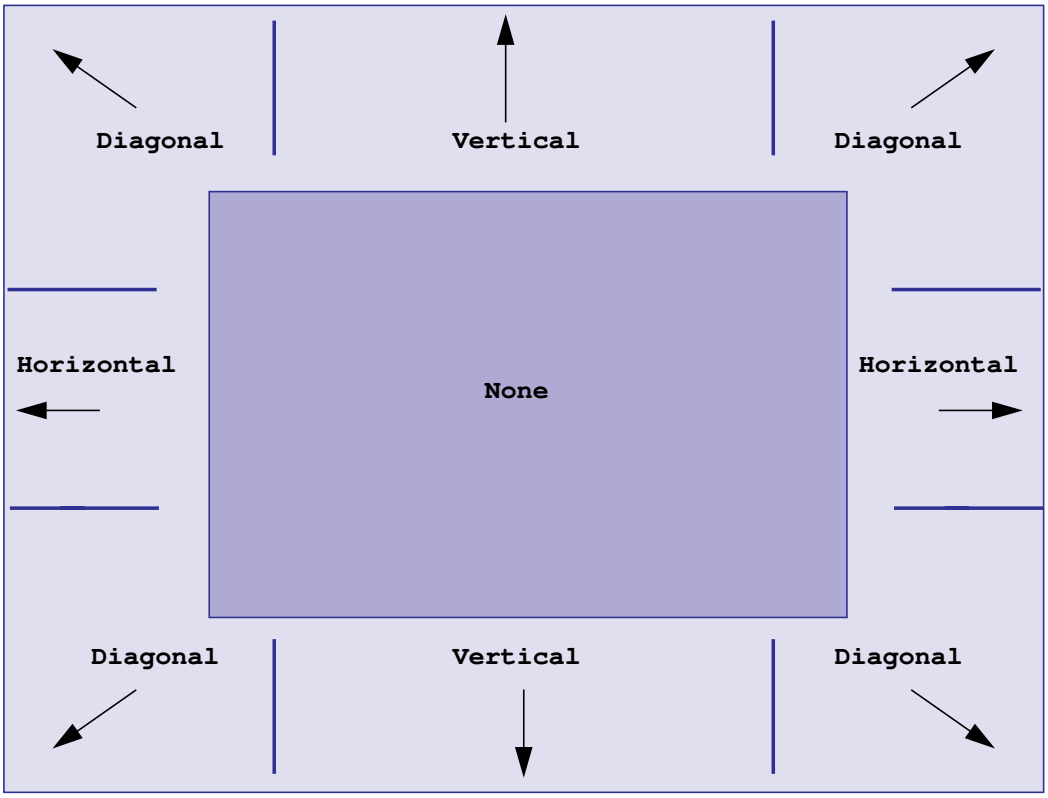
For an explanation of the documentation conventions used in this workbook, refer to the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Installation Procedure

For complete instructions on installing the data for this module, refer to "Installation Procedure" in the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Lesson 1
LAYOUT Graphics

The LAYOUT tool has a different set of graphic responses and layers than do the LIBRARIAN and FabLink tools. This lesson teaches you about the LAYOUT Edit window and how to move around your design.



Objectives

This lesson provides an introduction to the graphic features in the LAYOUT tool. The graphic features of the LAYOUT tool are as follows:

- Dynamic pan and zoom.
- Viewing by object.
- Reduced layer set.
- Additional colors and patterns.
- Smart guides.
- Viewing drill holes in pads and vias.
- Edit layer toggling.
- Improved routing grid display.

Dynamic Pan

The LAYOUT Edit window is divided into nine zones for panning the view. Panning occurs depending on where the cursor is in the Edit window.

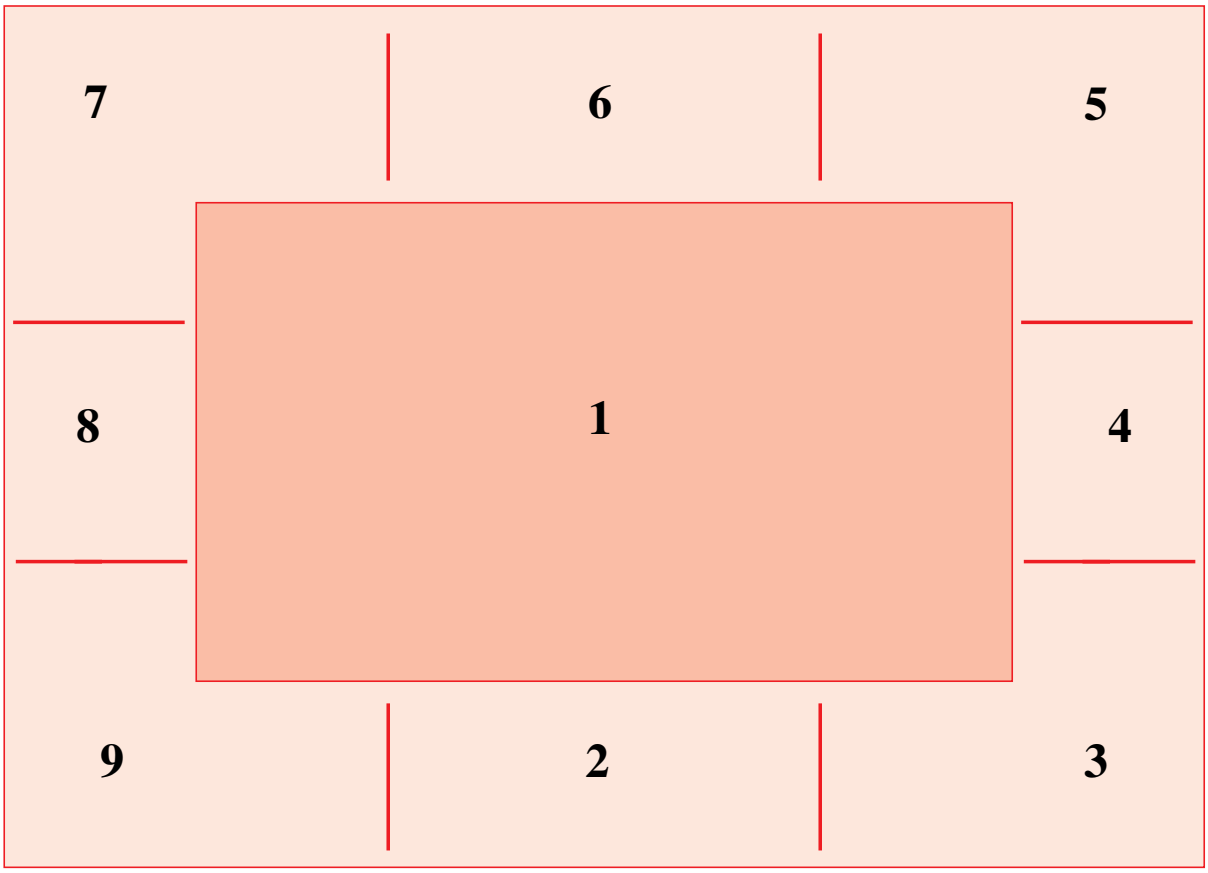


Figure 1-1. Panning Zones

- If the cursor is in zone 1, no panning takes place.
- If the cursor is in zones 2, 4, 6, or 8, the view pans orthogonally.
- If the cursor is in zones 3, 5, 7, or 9, the view pans diagonally.
- If the cursor is outside the board extent, no panning takes place.
- If you are viewing the entire board, no panning takes place.

Panning Zones

The following figure illustrates the panning zones of a board.

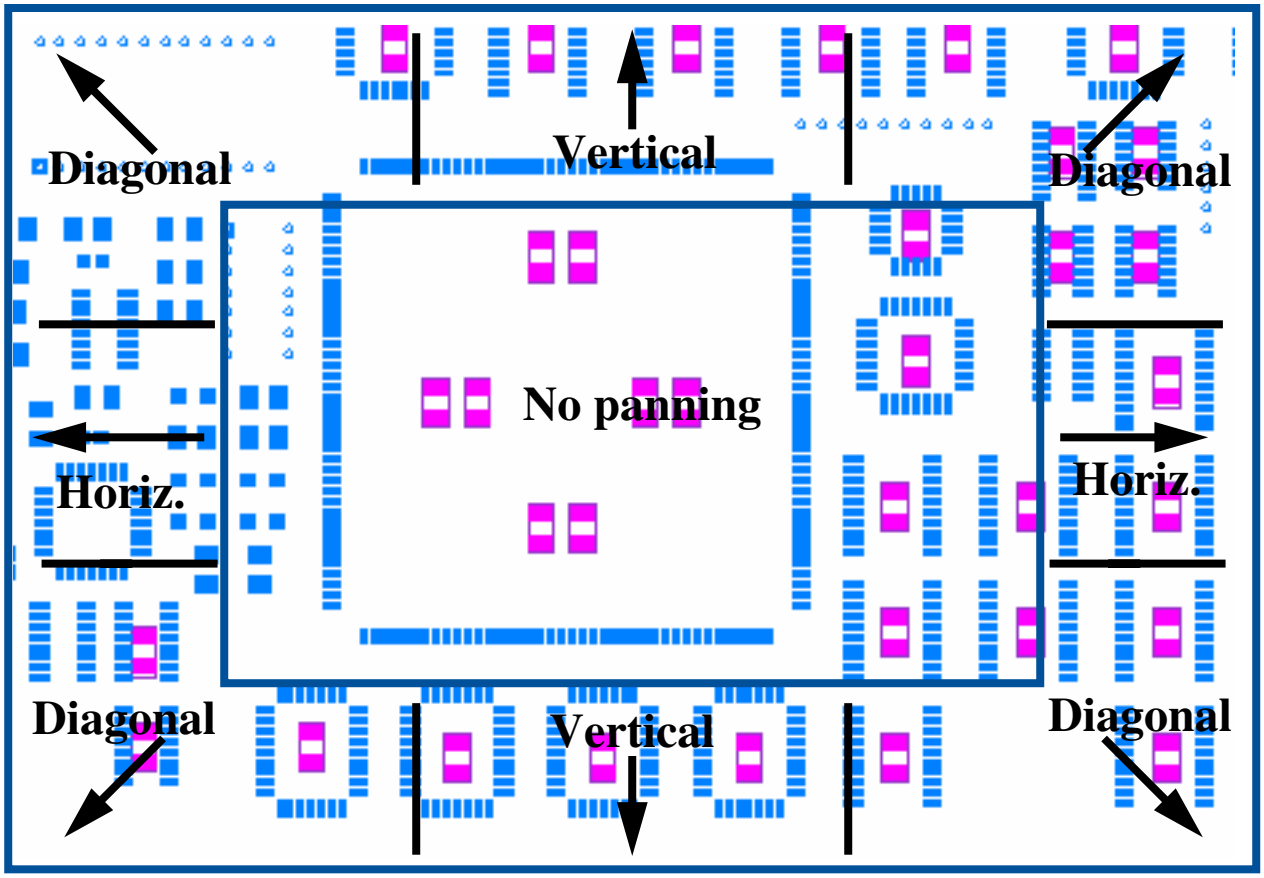


Figure 1-2. Zones Overlaid on a Board

The view pans automatically when the cursor pauses in an area of the Edit window defined as a pan zone. During panning, the view in the Edit window scrolls horizontally, vertically, or diagonally from the cursor location to bring another area of the board into view.

Once panning begins, the scrolling action continues until:

- You move the cursor out of the pan zone.
- The edge of the board comes into view.
- The pin to which you are interactively routing using the smart guide option comes into view.

Panning Speed

Panning speed depends on where the cursor is in a zone. Panning is fastest when the cursor is close to the outer edge of a zone. The darker shaded areas at the outer edges of the panning zones illustrate where to place the cursor in the zones for the fastest panning.

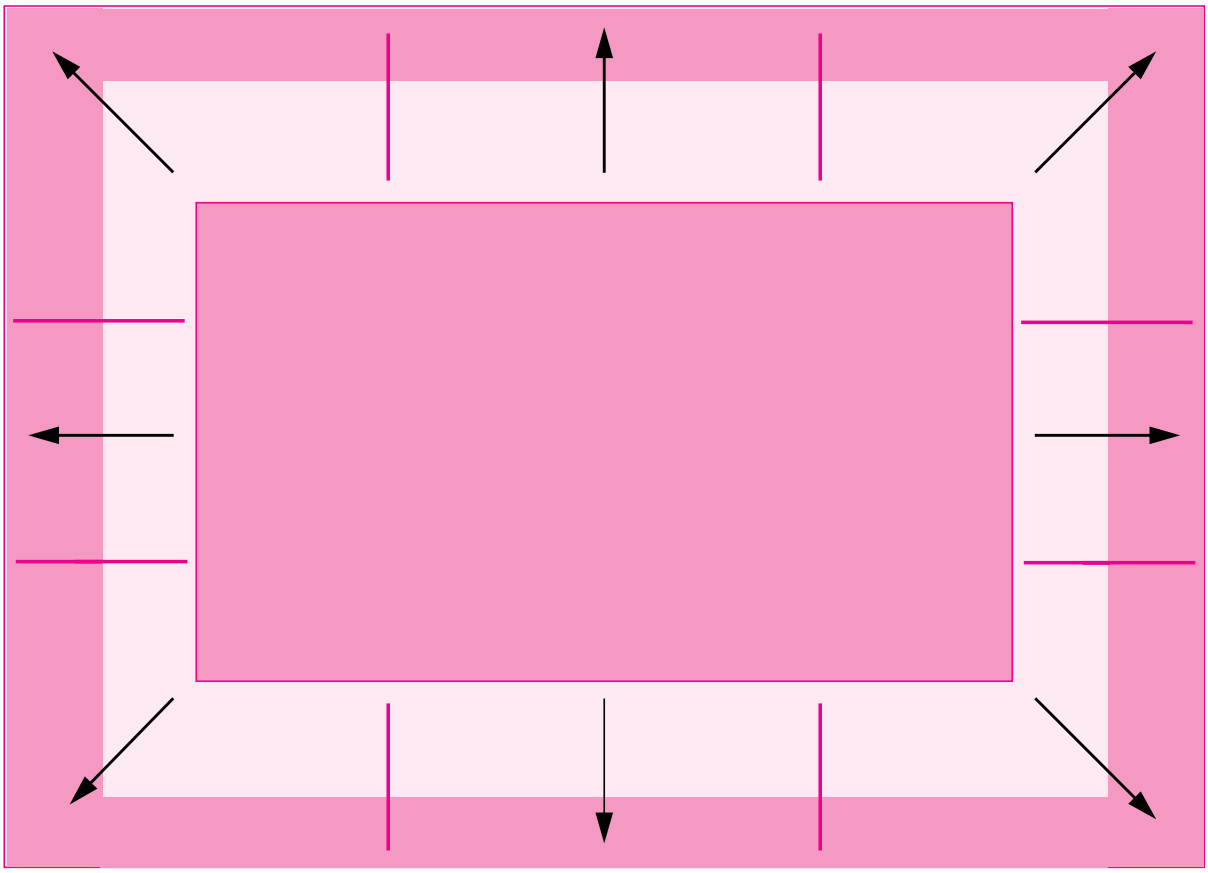


Figure 1-3. Outer Edge for Faster Panning

You can control the width of the pan zone around the border of the Edit window and the portion of each side of the pan zone border that is devoted to horizontal and vertical panning. Two other settings related to panning are pan delay and pan speed. Pan delay is the length of the interval between the pause of the cursor inside the pan zone and the start of panning. Pan speed is the rate at which the view scrolls. You learn more about these controls later in this lesson.

Dynamic Zoom

For dynamic zoom, the cursor can be anywhere in the Edit window. To zoom in, press and hold CTRL and the middle mouse button. To zoom out, press and hold SHIFT and the middle mouse button.

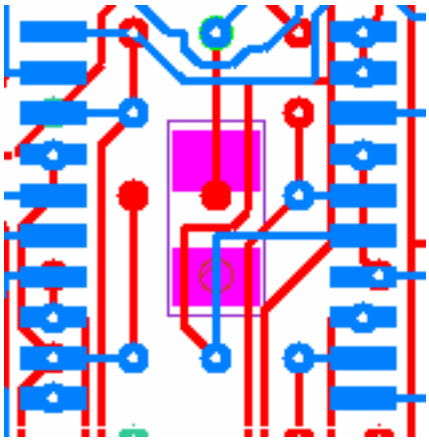


Figure 1-4. CTRL—Middle Mouse Button to Zoom In

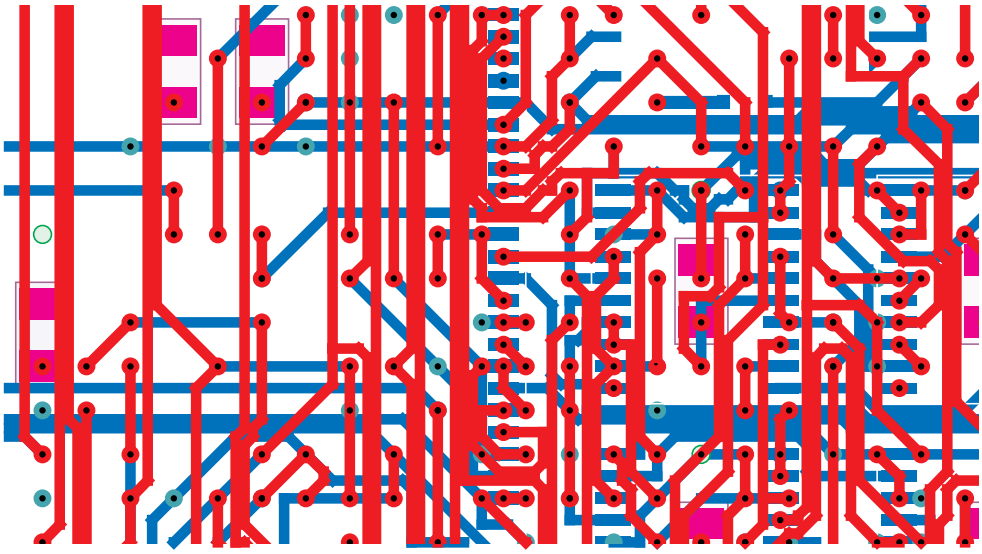


Figure 1-5. SHIFT—Middle Mouse Button to Zoom Out

You can control the settings for zoom delay and zoom speed.

Setting Pan and Zoom Parameters

You can set up parameters to control panning and zooming using the **Setup > Pan and Zoom** menu item.

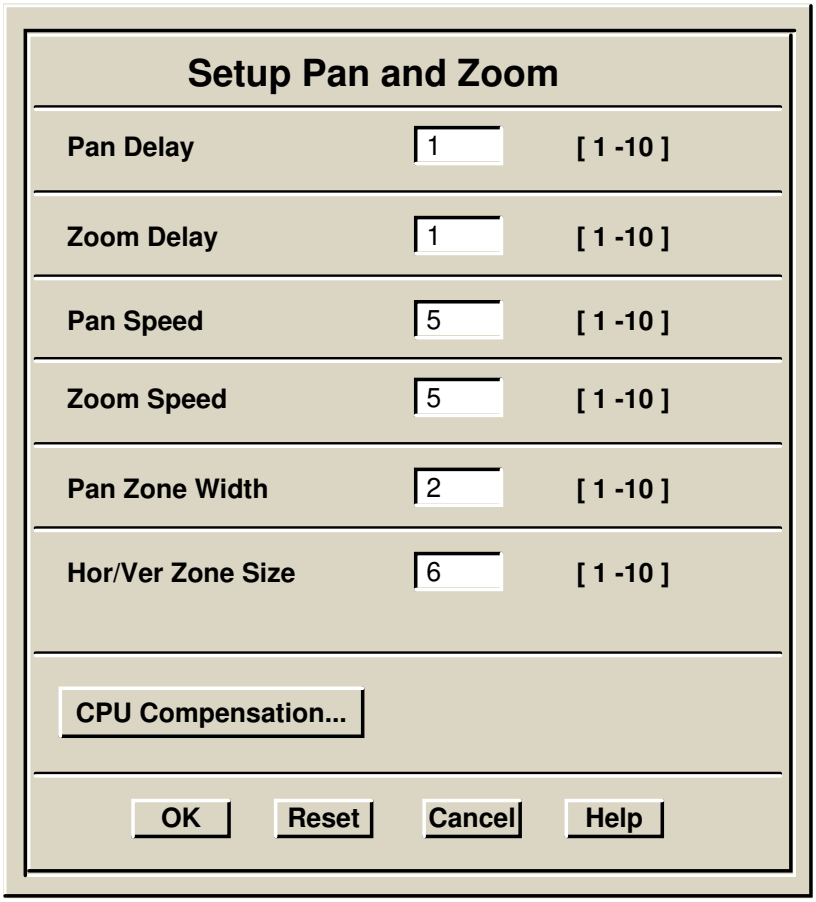


Figure 1-6. Setup Pan and Zoom Dialog Box

Pan delay is the length of time between the pause of the cursor in the pan zone and the start of panning.

Zoom delay is the length of time between the initiation of zooming and the onset of zooming.

Pan speed determines the fastest panning speed that occurs when the cursor is at the outer edge of a panning zone.

Zoom speed is the speed at which zooming occurs.

Setting Zone Size

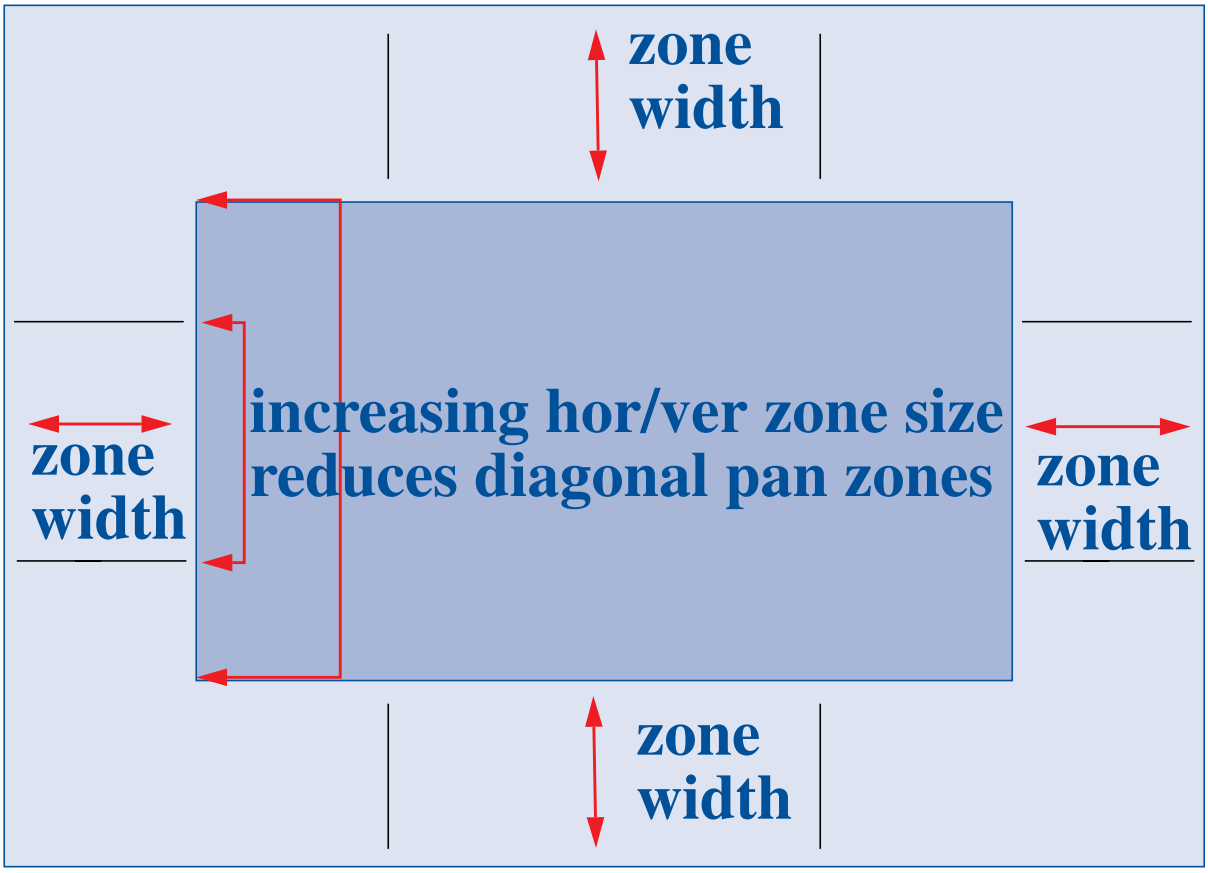


Figure 1-7. Zone Size Settings

Pan Zone Width is the width of the panning area band around the inside of the Edit window.

Hor/Ver Zone Size is the portion of each side of the pan zone that is designated for orthogonal panning. Increasing the size of the horizontal and vertical pan zones reduces the size of the diagonal pan zones. Reducing the size of the horizontal and vertical pan zones increases the size of the diagonal pan zones.

LAYOUT Layers

The LAYOUT tool has a reduced layer set as a result of adding the objects for controlling the display of graphical data. In addition, LAYOUT does not contain some of the generic layers found in LIBRARIAN and FabLink. The following layers are not available in the View Layers dialog box in the LAYOUT tool:

- Breakout, Breakout_1, Breakout_2
- Dam, Dam_1, Dam_2, Dam_3
- Density, Density_1, Density_2, Density_3, Density_4, Density_5
- Drawing, Drawing_1, Drawing_2
- Drill, Drill_holes
- Error
- Force
- Guide
- Histogram
- Pad, Pad_1, Pad_2
- Pin_id_1, Pin_id_2
- Probe, Probe_symbol
- Route_grid, Route_grid_1 through Route_grid_12
- Solder_mask
- Via, Via_usage

If you bring an existing design into LAYOUT with data on any of the layers in the Drawing, Breakout, and Dam layer sets, the layers containing the data are included in the View Layers dialog box.

LAYOUT Data Layers

LAYOUT stores graphical data on the following types of layers.

- Signal layers
- Power layers
- Graphical attribute layers
- Manufacturing data layers
- User-defined layers

You control whether a particular layer is visible or invisible. A layer must be visible in order to see the visible objects on the layer. You also control whether the visible objects on a layer are selectable or unselectable.

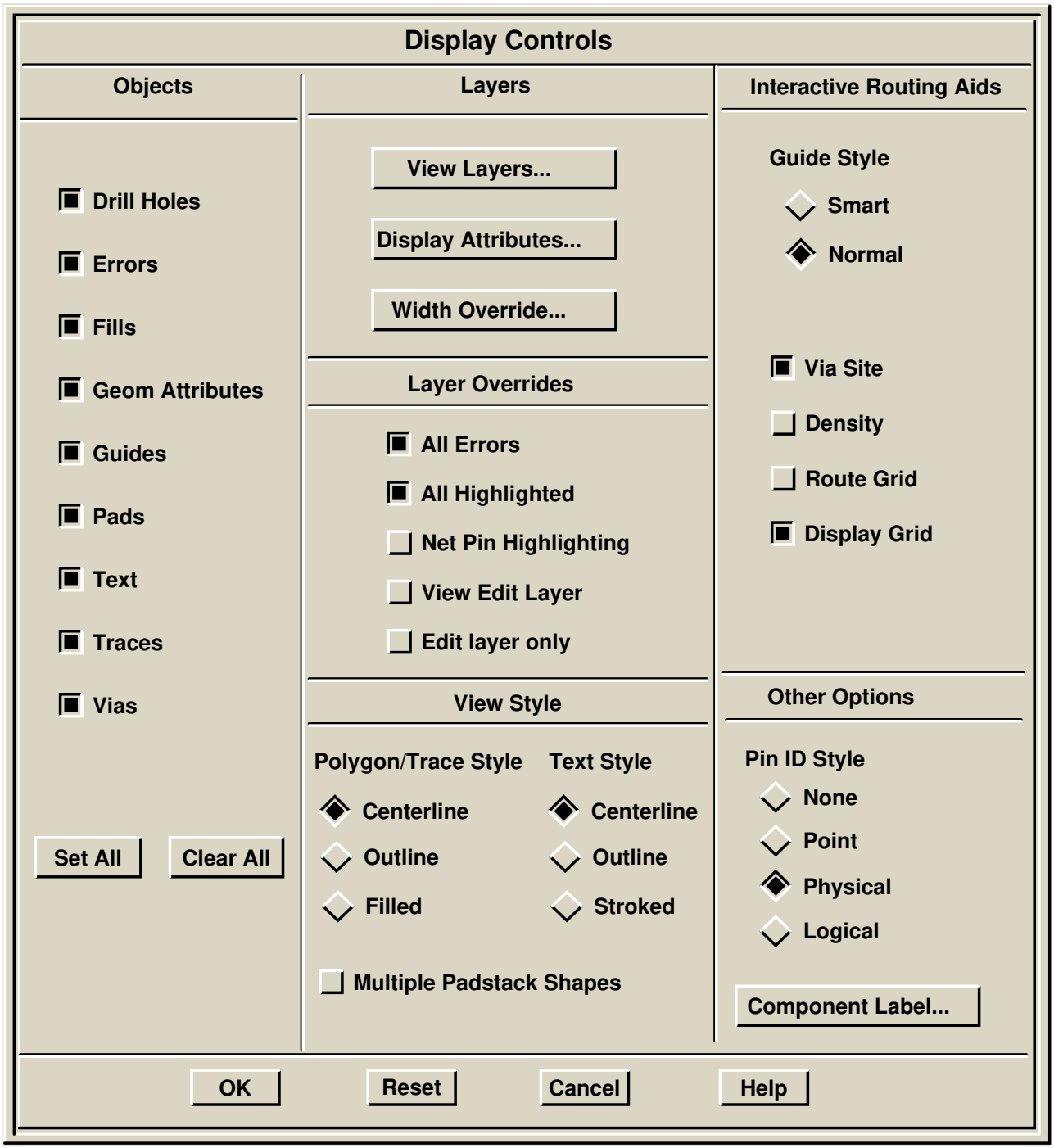
LAYOUT Objects

LAYOUT categorizes the following types of graphical data as objects.

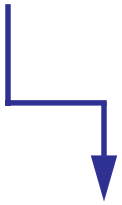
- Component labels
- Density
- Display grid
- Drill holes
- Errors
- Fills
- Geometry attributes
- Guides
- Pin pads
- Pin Identifications (Pin Id)
- Route Grid
- Text
- Traces
- Via sites
- Vias

Display Controls for Layers and Objects

You control the visibility of objects using the Display Controls dialog box.

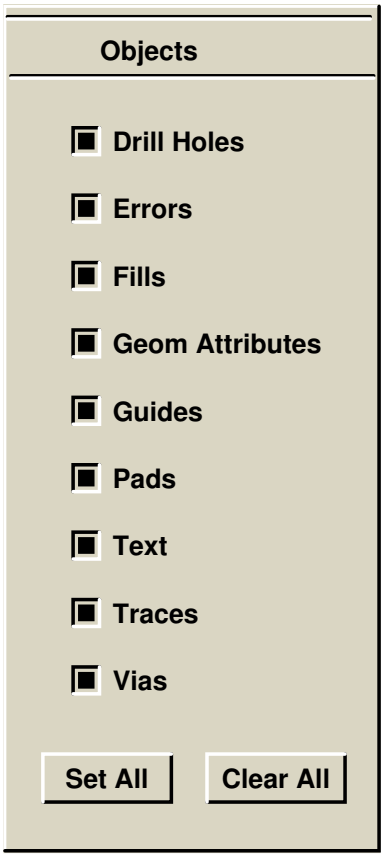


Displaying the Display Controls Dialog Box



You can display the Display Controls dialog box by selecting the menu item **View > Objects** or by using the middle mouse button to draw the View Objects Stroke, shown to the left.

Viewing Objects



Viewing Objects acts as a filter to determine what objects you view on a specific layer. View Objects works in conjunction with View Layers. For example, you want to view your traces on the top side of the board. In the View Layers dialog box, you select to view layer Signal_1. Layer Signal_1 is now visible. If you do not also select to view the Traces object in the Display Controls dialog box, you would see other objects on layer Signal_1, but not traces.

The filtering applies to Fills, Geom Attributes, Pads, Text, Traces, and Vias.

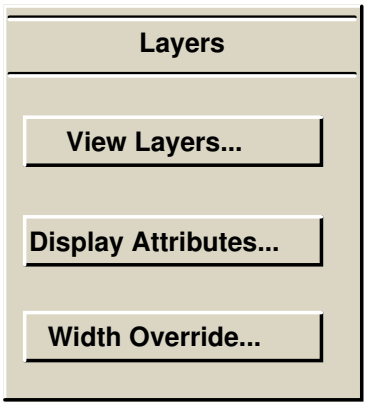
Viewing Padstacks

The Drill Holes and Pads objects control how you view padstacks.

Selecting Drill Holes allows you to view the drill holes in pads and vias. Viewing drill holes other than those in pads and vias is determined by the visibility of the Drill and Drill_holes layers.

Selecting the object Pads displays both through-pin and surface-mount pads. Surface-mount pads display if the layer on which a pad shape exists is also set as visible.

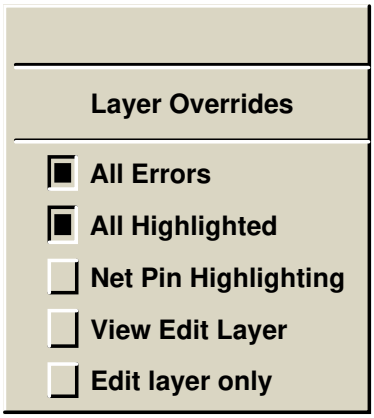
Layers



The **Layers** section of the dialog box provides access to the **View Layers** dialog box for setting the visibility and selectability of layers. It also provides access to the **Set Display Attributes** dialog box for setting the colors and patterns of layers and objects.

Select the **Width Override** button to open the Set Object Width Override dialog box. By typing a **Width Override** value from 0-5 in the dialog box, you can change the display for selected, protected, and highlighted objects and for errors. A zero value for the **Width Override** is equivalent to the default condition, which uses the colors and patterns assigned in the Set Display Attributes dialog box for their display. By setting the **Width Override** to a value from 1-5, you change the display of selected/protected/highlighted objects and errors to an outline in the assigned color without the display of the pattern. A value of 1 results in a narrow outline; a value of 5 gives a wide outline. After setting the width override value, select **OK** to execute the dialog box and return to the Display Controls dialog box.

Layer Overrides



The **Layer Overrides** area of the dialog box allows you to change the display by superseding the current visibility settings of objects and layers.

The **All Errors** button displays all of the objects within the current view that are in error, regardless of the visibility settings for objects and layers.

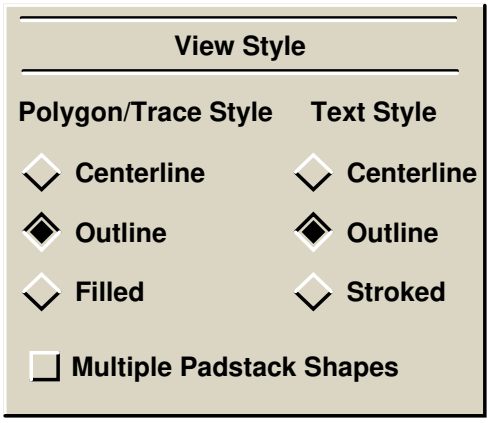
The **All Highlighted** button displays all highlighted objects within the current view, regardless of the visibility settings for objects and layers.

The **Net Pin Highlighting** button highlights the pin padstacks associated with the selected net.

The **View Edit Layer** button enables refreshes of the Edit layer each time changes are made to this layer. For denser designs, you can set this switch to off to reduce the number of automatic refreshes during interactive routing.

The **Edit layer only** button displays only the objects on the current edit layer. This selection turn off the display of all other signal and power layers regardless of their visibility setting. When the display is set to **Edit layer only**, the selections of **All Errors** and **All Highlighted** have no effect.

View Style



You choose from the display styles under the **Polygon/Trace Style** or **Text Style** label. The choices for Polygon/Trace are **Centerline**, **Outline**, and **Filled**.

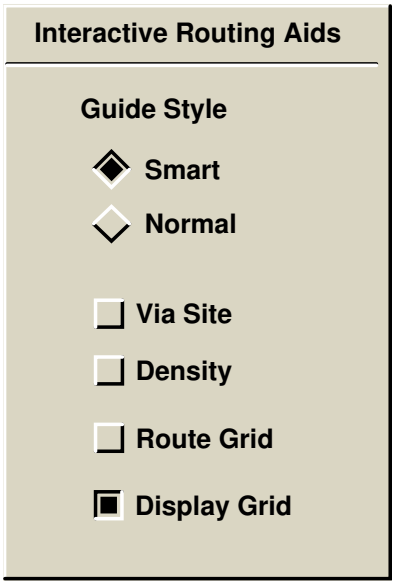
- **Centerline** displays polygons and graphical elements as outlines and displays traces as centerlines.
- **Outline** displays polygons and graphical elements as outlines and displays traces with their actual width.
- **Filled** displays polygons and graphical elements with a fill pattern and displays traces with their actual width and a fill pattern.

The choices for Text are **Centerline**, **Outline**, and **Stroked**.

- **Centerline** displays text as centerlines.
- **Outline** displays text with its actual width.
- **Stroked** displays text as an outline showing the stroke width if the stroke width is greater than zero.

Select the **Multiple Padstack Shapes** button to display pin and via padstack shapes on all the visible layers. Unselect this button to display only the padstack shape on the edit layer or, if a padstack shape does not occur on the edit layer, the padstack shape on the next visible layer.

Interactive Routing Aids



The **Interactive Routing Aids** area of the dialog box provides some display alternatives that are useful during interactive routing.

Under the **Guide Style** label are two options for controlling the display of guides. The **Guides** object must be set to visible for these options to have an effect. Select one of the options.

Select the **Smart** button to display only the guides of the net that is currently selected for routing. When you select a guide or vertex to route, the guides that are attached to all other nets on the board disappear from the display. When the connection is complete, all remaining guides reappear.

Select the **Normal** button to maintain the display of guides for all nets during interactive routing.

Select the **Via Sites** button to display the possible via locations around the cursor location. The via sites that display are always on the via grid. The maximum number of via sites in the display is nine.

Display Density

Select the **Density** button to see visual feedback on the density of the currently visible objects on the board. The density information appears as ten shades of the density color, with the lightest shade showing the least dense regions and the deepest shade showing the most dense regions.

LAYOUT calculates density based on viewed objects. The greater the density of the viewed objects in a specific area of the board, the brighter the shading in that area.

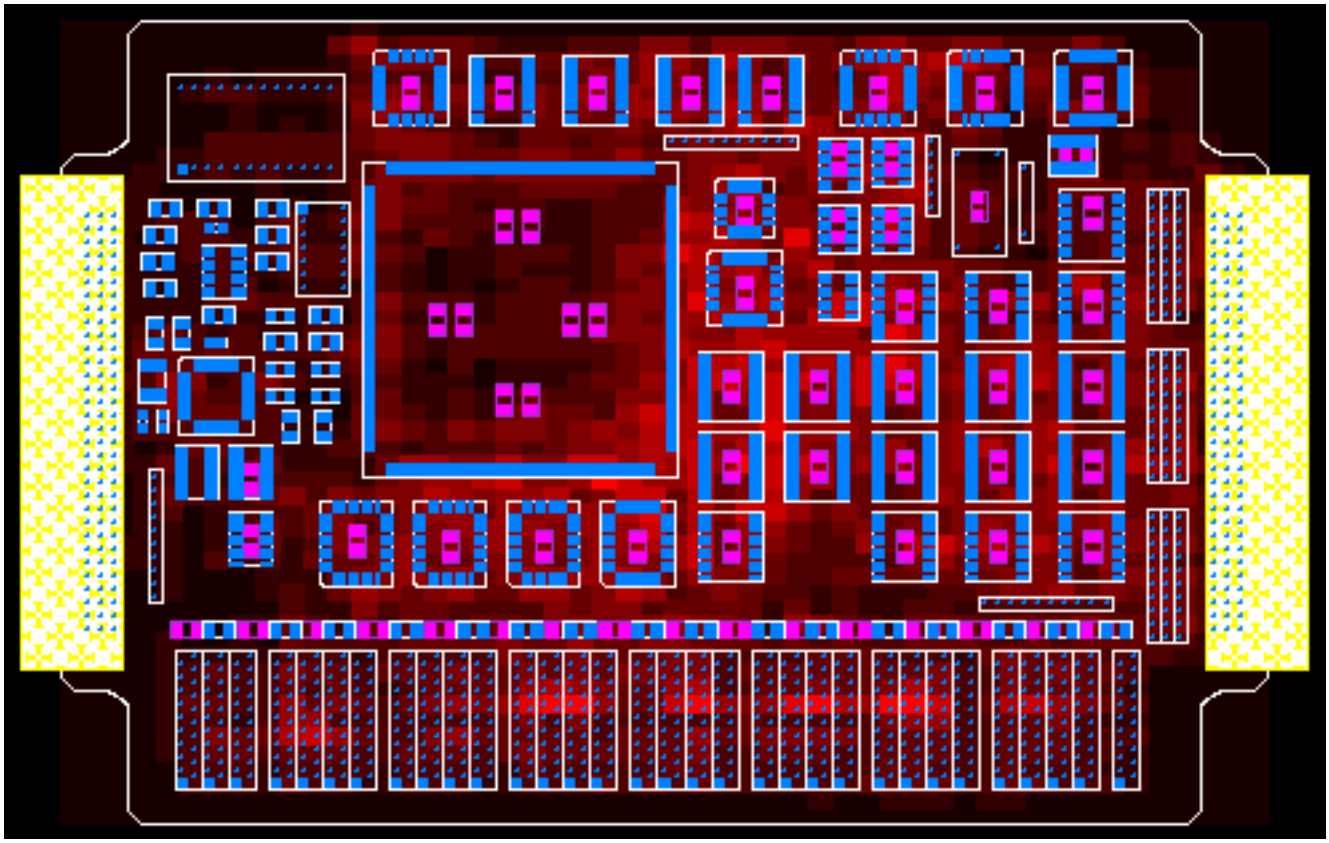


Figure 1-8. Board Density Displays as Shading

Displaying Grids

Route Grid

LAYOUT now treats the routing grid as an object as it does pads and drill holes. You select to view the routing grid in the View Objects dialog box. Whether or not you see the grid depends on how far you zoom in on the board. If the grid is too dense, the routing grid does not display.

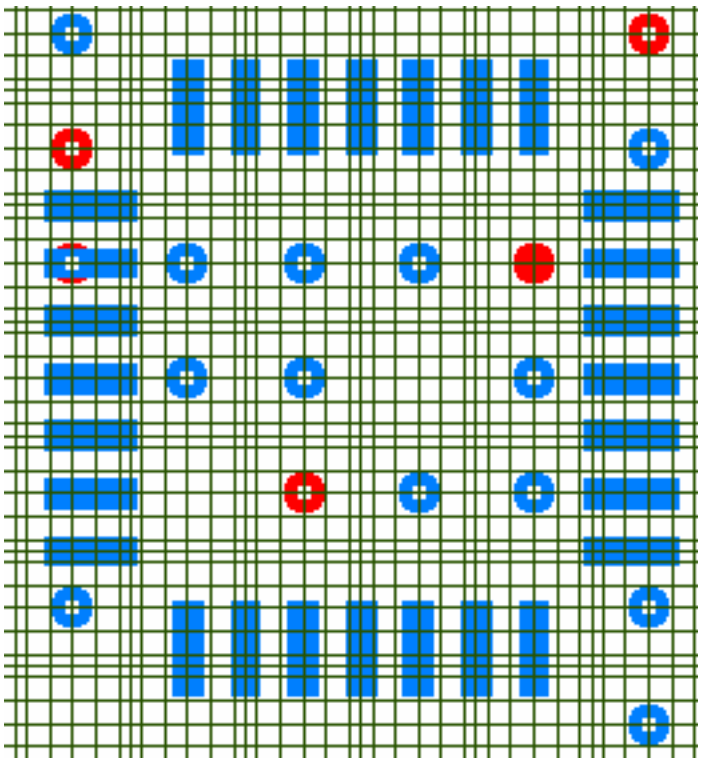
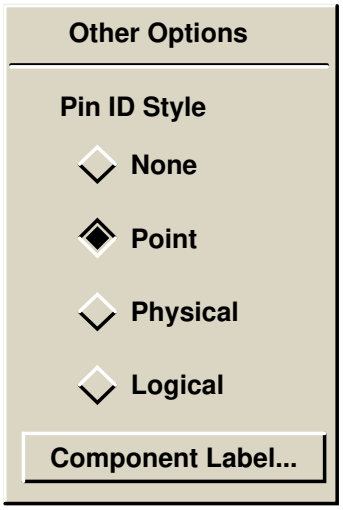


Figure 1-9. Routing Grid Displayed

Display Grid

Select the **Display Grid** button to see the reference grid in the Edit window.

Other Options



The **Other Options** area of the dialog box controls the display of pin IDs and component labels.

Under the **Pin ID Style** label are the options for controlling the display of pin IDs. Select one of the options.

Select the **None** button to remove pin IDs from the display.

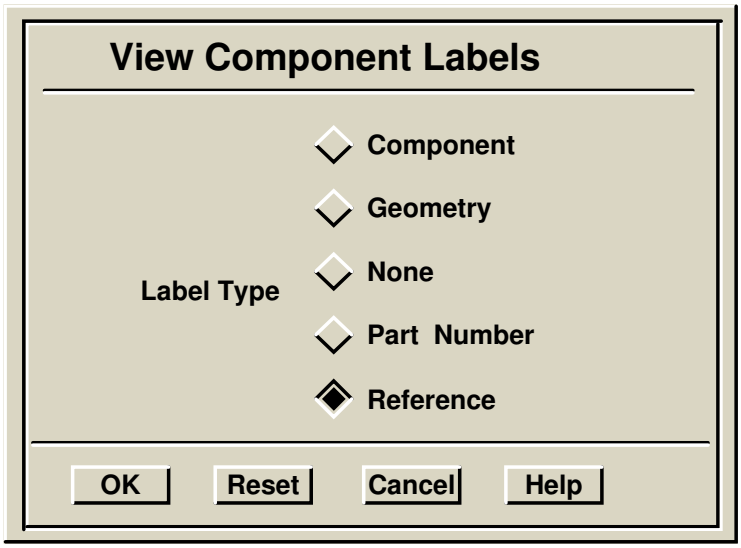
Select the **Point** button to display a small square for the pin ID.

Select the **Physical** button to display the physical pin number of the pin.

Select the **Logical** button to display the logical pin name of the pin.

The physical pin number and the logical pin name display as text, so the Text object must be set to visible if you select either the **Physical** or **Logical** pin ID style. If the pin number/ pin name text is too small to display in the current view, the pin ID displays as a point.

Component Labels



Select the **Component Label** button for direct access to the View Component Labels dialog box. The dialog box offers a choice of options for displaying informational text that is assigned to components. Select one of the **Label Type** options. The selected label displays on all components that carry a label of the selected type. The component labels display as text, so the Text object must be set to visible to enable the label display.

Select the **Component** button to display the component type label. For example, 74LS374.

Select the **Geometry** button to display the component geometry name. For example, dip20.

Select the **None** button to remove component labels from the display.

Select the **Part Number** button to display the part number of the component. For example, pn-decoupler or pn-002.

Select the **Reference** button to display the reference designator of the component. For example, U23.

Select the **OK** button to execute the View Component Labels dialog box and return to the Display Controls dialog box.

Set Display Attributes

For layers, as well as object types **Drill Holes**, **Errors**, **Guides**, **Pads**, and **Vias**, you can set the colors and patterns for their display through the Set Display Attributes dialog box.

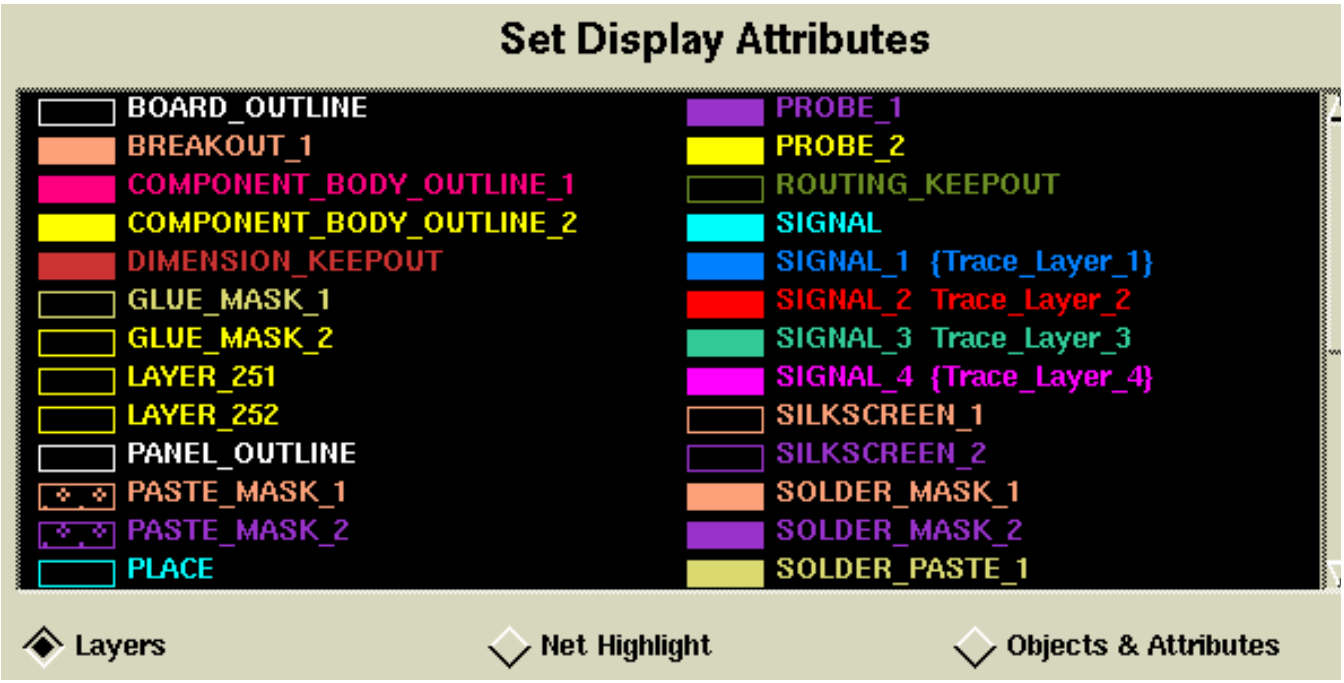


Figure 1-10. Set Display Attributes Layers and Objects

The object types of **Fills**, **Text**, and **Traces** display in the color and pattern of the signal or power layer on which they reside.

The **Geom Attributes** objects display in the color and pattern of their associated layer.

The **Errors** object displays the errors related to visible objects on visible layers.

Color and Pattern Display Attributes

The Set Display Attributes dialog box allows you to choose from many colors and patterns. You can assign the colors and patterns to both layers and objects.

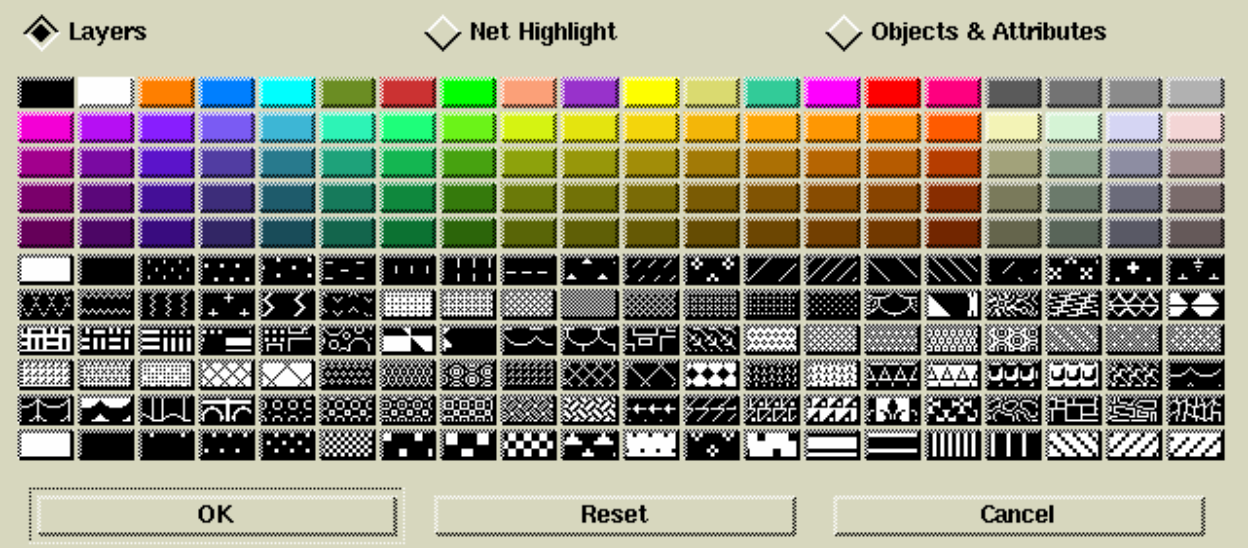


Figure 1-11. Set Display Attributes Colors and Patterns

Setting Net Highlighting

The **Net Highlight** button allows you to differentiate between nets by applying colors and patterns used when highlighting specific nets.

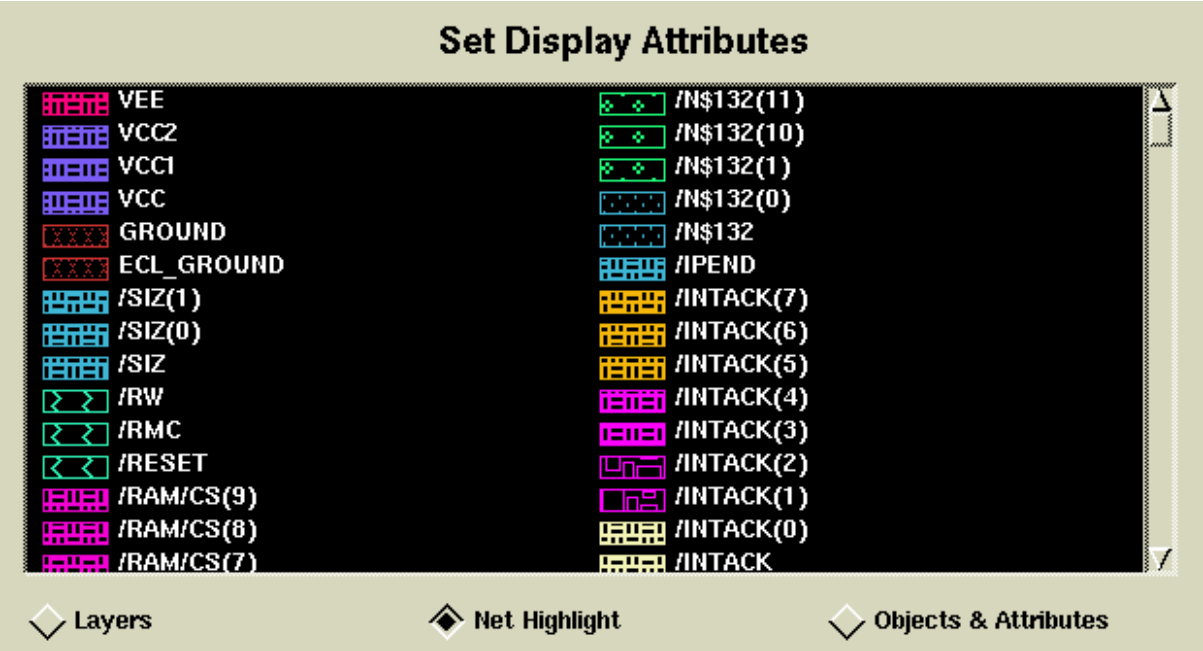


Figure 1-12. Determine Net Highlighting

The **Net Highlight** option allows you to override the default color and pattern for highlighting individual nets, as set under the **Objects & Attributes** option. In front of each net name is a box that shows the current color and pattern assigned as the net highlight.

When you highlight a net in the Edit window, the highlighting shows the traces, vias, pins, and area fills of the net.

Setting Additional Highlighting

Selecting the **Objects & Attributes** button allows you to set additional highlighting attributes.

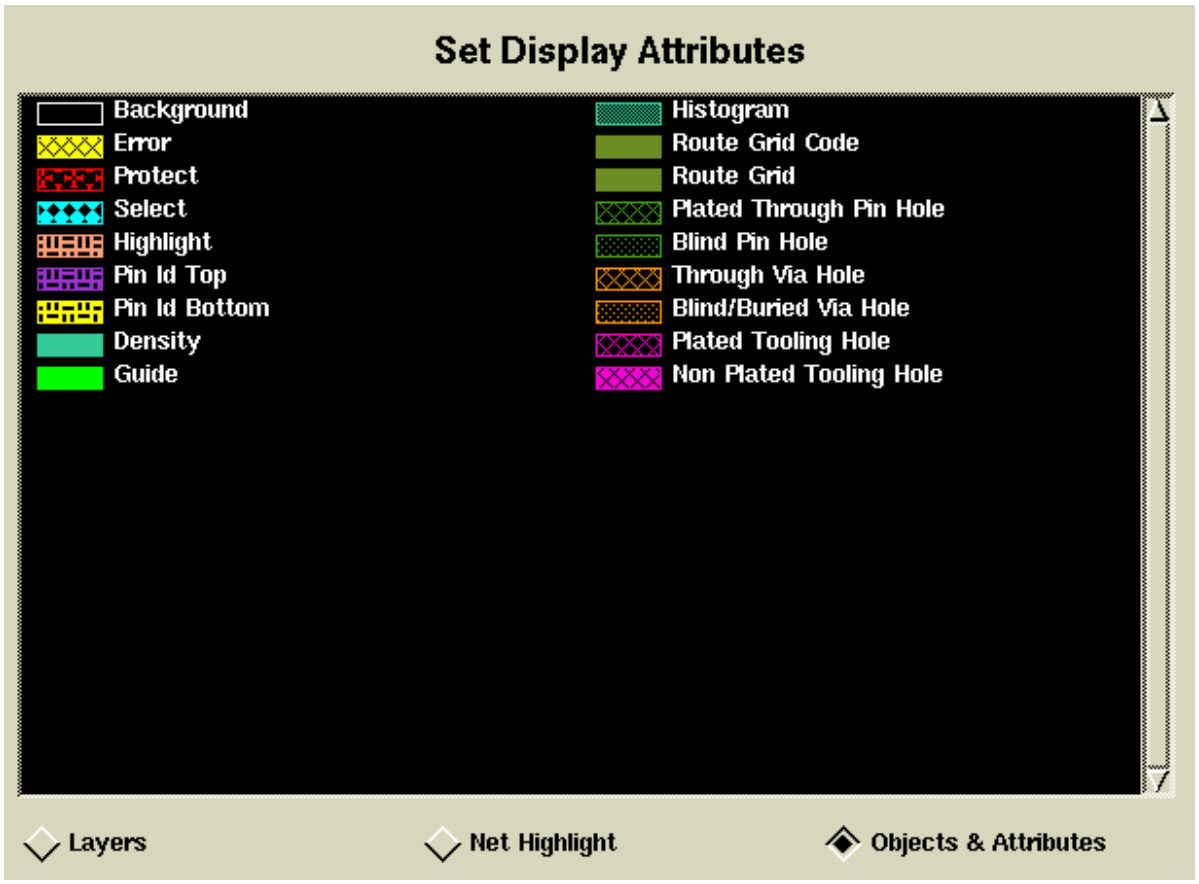


Figure 1-13. Determine Additional Display Attributes

Additional information about the settings for **Objects & Attributes** is provided in the following object and attribute descriptions.

Background—The background setting establishes the background color in the Edit window. You can choose any of the 100 colors for the background; however, the background does not accept a pattern assignment. The background of the lists in the Set Display Attributes dialog box always reflects the current Edit window background color.

Error, **Protect**, **Select**, **Highlight**—These settings establish the default colors and patterns for displaying objects that are in error, selected, protected, or highlighted. With the **Net Highlight** option in this dialog box, you can override the default color and pattern for net highlighting on a net-by-net basis. With the **Width Override** option in the Display Controls dialog box, you can override the use of the pattern for errors, selection, protection, and highlighting, even if the view style is set to filled.

Pin Id Top, **Pin Id Bottom**, **Density, Guide**, **Route Grid Code**, **Route Grid**—These settings control the display of objects that you make visible through the Display Controls dialog box. You set the color and pattern for these objects because these objects are not associated with any of the physical board layers. Objects that are a part of a physical layer display with the color and pattern assigned to the layer, as set under the **Layers** option of this dialog box.

Histogram—The histogram display attributes relate to the placement histogram, which you view by selecting **Report > Placement Histogram**.

Plated Through Pin Hole, **Blind Pin Hole**, **Through Via Hole**, **Blind/Buried Via Hole**, **Plated Tooling Hole**, **Non Plated Tooling Hole**—These settings control the display of drill hole, pad, and via objects and allow you to see and identify the various types of drill holes associated with the pin pads, vias, and tooling holes in your design.

Opaque Objects

In LAYOUT, all objects draw as opaque objects. This means objects on top partially hide objects underneath. The Edit layer is the layer on the top.

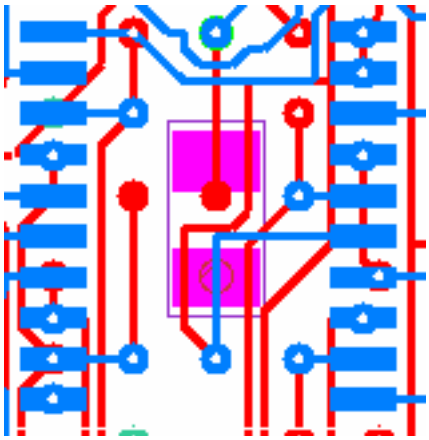


Figure 1-14. Layers Display as Opaque

You can click the Select mouse button in the Edit layer display area to cycle through the current set of Edit layers. In turn, objects on each Edit layer pop to the top. You define the set of Edit layers by choosing the **Setup Routing > Edit Layer** menu item.

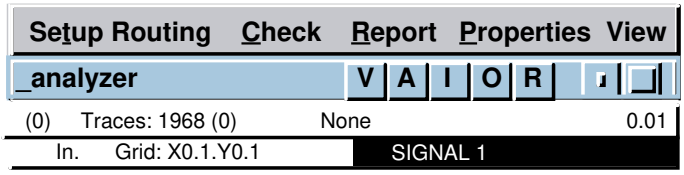


Figure 1-15. Edit Layer Display Area

You can view objects behind other objects by displaying layers in patterns other than solid. The Set Display Attributes dialog box allows you to choose from many new colors and patterns.

Lab Exercise

In this lab exercise, you manipulate the LAYOUT graphics.

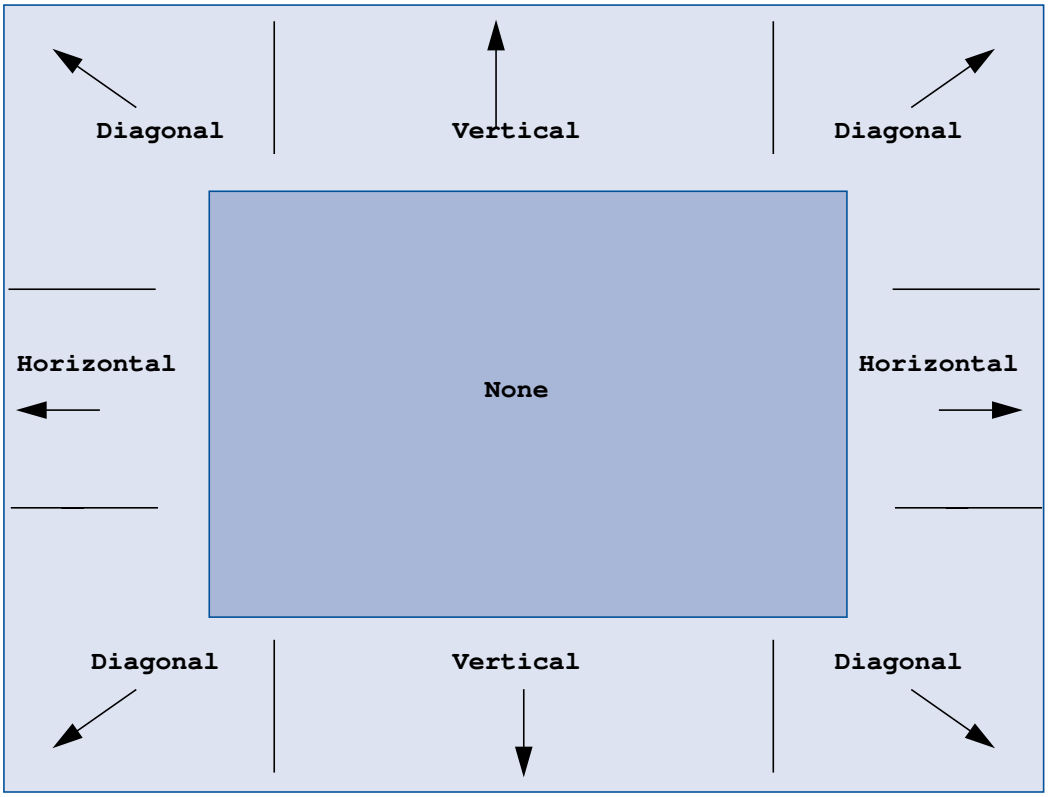
Upon completion of this lab exercise you can:

- Pan and zoom the Edit window.
- Set pan and zoom parameters.
- Set display attributes.
- View by object.
- View by component labels.

Turn to Module 5—Lab 1: “Layout Graphics".

Lab 1
LAYOUT Graphics

The LAYOUT tool has a different set of graphic responses and layers than do the LIBRARIAN and FabLink tools. This lesson teaches you about the LAYOUT Edit window and how to move around your design.



Introduction

In this lab exercise, you learn how to pan and zoom around your board. You also learn to control the display of layers and objects.

Upon completion of this lab exercise you should be able to:

- Dynamically pan and zoom.
- View by object.
- Set additional colors and patterns.
- View drill holes in pads and vias.

Procedure

You use LAYOUT to pan and zoom around your board.

Preparation for Lab

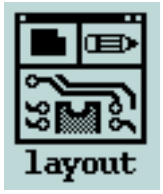
To set up for this lab you need to invoke the LAYOUT tool on your design.

1. If you or your instructor have not already done so, complete the Installation Procedure in the "About This Training" section of Module 1 of the *Board Station for New Users Training Series*.

2. Invoke the Design Manager by entering the following in a shell:

   Sys V> ***$MGC_HOME/bin/dmgr***

3. Using the Design Manager, change your current directory to the board_new directory by clicking on the four-way icon in the navigator window. In the **Change directory to** dialog box, enter the pathname: your_path/**training/board_new/mod5** and press the Return key.



4. Find the LAYOUT icon in the Tools window. Invoke LAYOUT by placing the cursor on the LAYOUT icon and double clicking the Select mouse button.

   The INVOKING LAYOUT: Select a Design dialog box displays.

5. Select the **sig_az** design and **OK** the dialog box.

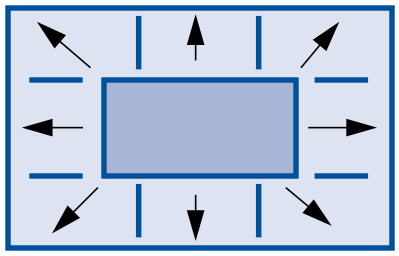
   If an Invocation Switch dialog box appears, select the **OK** button without selecting the switch.

   A Report-Startup message appears in the middle of the LAYOUT Session window. This is a list of notes concerning the files used to invoke the LAYOUT tool. After reading the report notes, **Close** the report window, and then maximize the size of the LAYOUT session window to fill the display.

Panning and Zooming the Display

1. Zoom in on the board, by pressing and holding CTRL-Middle Mouse Button.

2. Pan the Edit window view by moving the cursor to various zones in the Edit window.



Note where your cursor is within a zone and how cursor placement affects panning speed.

3. Zoom out using SHIFT-Middle Mouse Button.

Changing Pan and Zoom Settings

1. Select menu item **Setup > Pan and Zoom** to display the dialog box.

2. Leave **Pan delay** and **Zoom delay** each set to 1.

3. Set the **Pan speed** and the **Zoom speed** each to 10. **OK** the dialog box.

4. Zoom in on the board and note the increase in zooming speed. Also move the cursor into various pan zones and note the difference in panning speed. Test any other speed you like, as well.

   Allowable Pan speed and Zoom speed settings are between 1 and 10. The default setting for both Pan speed and Zoom speed is 5.

Changing the Pan Zones

1. Using the Setup Pan and Zoom dialog box, increase the **Pan zone width** to 6. **OK** the dialog box.

2. Move the cursor between the non-panning zone in the middle of the board and one of the panning zones. Note the size increase of the panning zones.

   The default Pan zone width setting is 2.

   Now change the size of the diagonal panning zones.

3. Using the Setup Pan and Zoom dialog box, change the **Hor/Ver zone size** to 10. **OK** the dialog box.

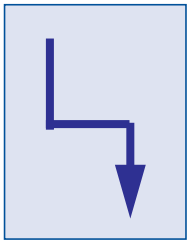
4. Test the results by panning the display in an orthogonal direction and then try to pan in a diagonal direction. Note that your diagonal panning zones have expanded while the orthogonal panning zones have decreased in size.

   The default Hor/Ver zone size setting is 6.

5. Use the Setup Pan and Zoom dialog box to return each setting to its default. Defaults are:

   Pan and Zoom speeds: 5
   Pan zone width: 2
   Hor/Ver zone size: 6

Setting Display Attributes



1. Choose the **View > Display Controls** menu item. Or, use the Display Controls stroke, shown at left, to display the dialog box.

2. In the Layers section of the dialog box, click on the **Display Attributes...** button.

   The Set Display Attributes dialog box displays. By default, the dialog box first gives you the list of layers to which you can assign colors and patterns.

   A two-column list of the layer names is contained in the dialog box, with each name appearing in the color in which graphics on that layer are created. Next to each layer name is a box with the color and pattern assigned to that layer.

3. Click on the **Net Highlight** button to see the list of nets in the design.

   You can assign display attributes to nets. When you highlight an individual net, the net highlights using the color and pattern you set.

4. Click on the **Objects and Attributes** button.

   You can change the color or pattern of any object or attribute in the list. You are going to change the color of the board background.

5. Click on **Background**.

6. Click on any color to make the assignment.

   The background of the list area changes to the color you select.

7. Note the color and pattern assigned to the **Protect** object. You are going to do something with protected objects in a moment. You can make changes to the assignment, if you wish, but ensure both a color and pattern are assigned to the Protect object.

8. **OK** the Set Display Attributes dialog box.

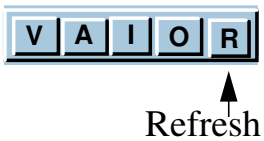
Viewing Protected Objects

1. Click on the **Width Override** button to display the Set Object Width Override dialog box.

2. Note that the width override is set to 0. A value of 0 displays protected objects as filled. But the protected components on the board are not displaying as filled. **Cancel** the Set Object Width Override dialog box.

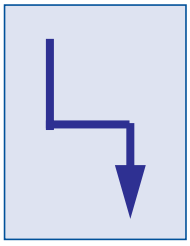
3. In the View Style section of the Display Controls dialog box, set the Polygon/Text/Trace Style to **Filled**. **OK** the Display Controls dialog box.

   The protected components now display filled in the assigned color and pattern.



In addition, the background of your board is now set to the color you selected. If needed, refresh the display by clicking on the *R* icon at the upper-right of the screen.

Changing the Width Override for Protected Objects



1. View the **Display Controls** dialog box. Choose the **Width Override** button.

2. Give the Width Override a value of **5** and **OK** the dialog box.

3. **OK** the Display Controls dialog box.

   Protected objects now display as thick outlines in the assigned color.

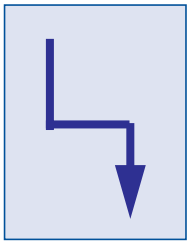
4. View the **Display Controls** dialog box. Choose the **Width Override** button.

5. Give the Width Override a value of **1** and **OK** the dialog box.

6. **OK** the Display Controls dialog box.

   Protected objects now display as thin outlines in the assigned color.

Controlling the Display



In this section, you set various controls to determine whether and how you view the layers and objects that make up your design.

1. View the **Display Controls** dialog box.
2. In the Objects section of the dialog box, select to view only the following objects:

   Drill_holes

   Geom Attributes

3. In the Layers section of the dialog box, click on the **View Layers...** button.

A *V* next to each name indicates visibility.

Next you set the view layers for the placement and routing keepout layers. These layers help you see legal component placement areas. Select to view only the following layers:

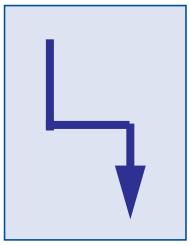
Board_outline, Placement_region_1

Place, Place_1, Place_2

If you cannot see one or more of the layer names, it is probably because the background color you set is not enough of a contrast to the layer color. If this is the case, change the background color back to Black.

4. **OK** the View Layers dialog box.
5. In the Other Options section, select **Component Label...**
6. In the View Component Labels dialog box, choose Label Type: **Reference**, and then **OK** the dialog box.
7. **OK** the Display Controls dialog box.

   Even though you specified to view the reference text, no text shows for the components.



8. Use the Display Controls stroke to display the dialog box.

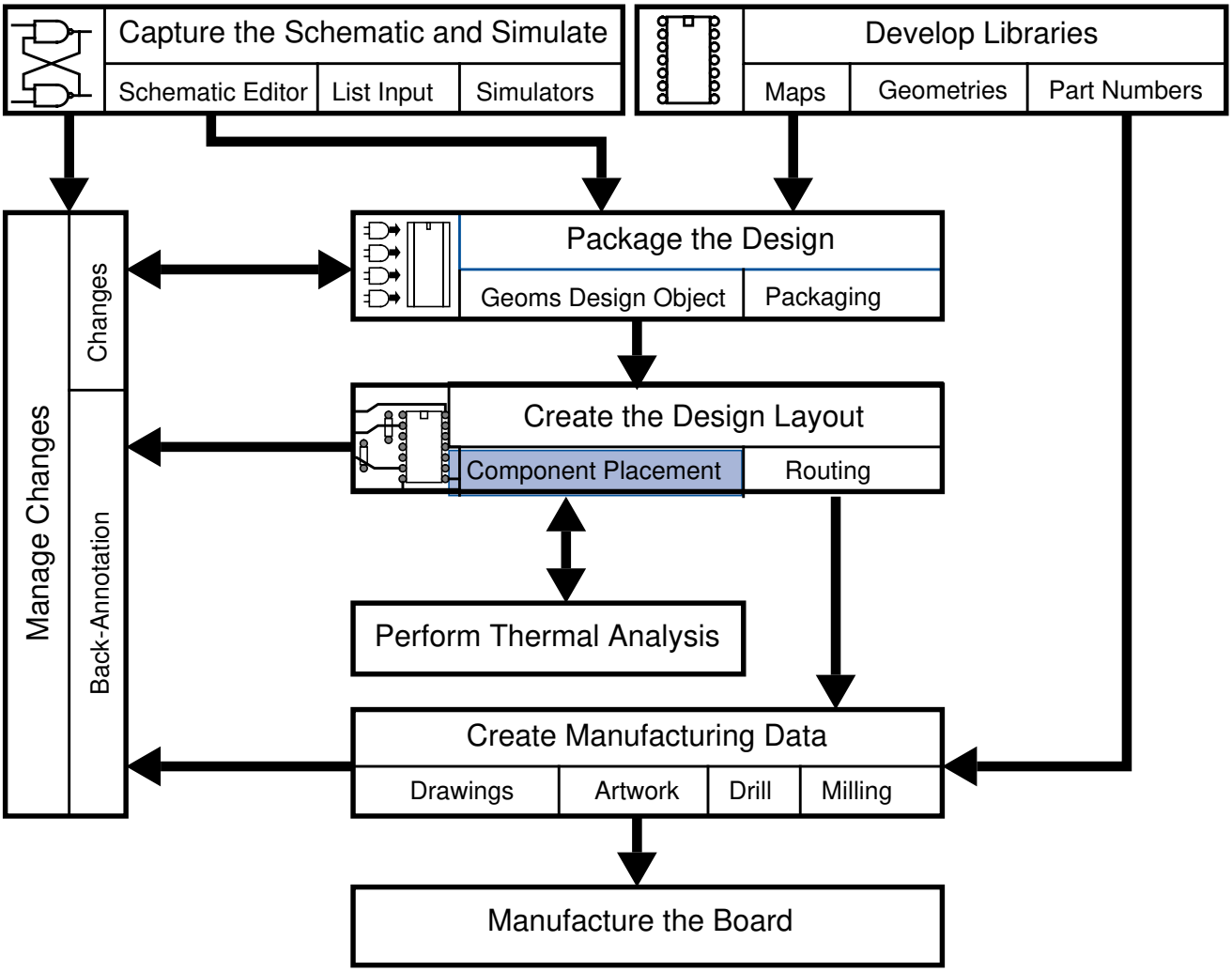
9. In the Objects section, add the **Pads** and **Text** objects to the list of viewed objects. **OK** the dialog box.

Now both the padstacks and the reference text show on the board. The reference text makes it easier to locate specific components when you begin placement.

Congratulations! You have completed the "LAYOUT Graphics" lab exercise. Throughout the placement and routing labs, you exercise additional features of the LAYOUT graphics package. Continue with Lesson 2: "Component Placement Setup".

Lesson 2
Component Placement Setup

The Placement and Routing modules provide an introduction to the major features of the LAYOUT tool. The Placement module acquaints you with the interactive and automatic placement features and prepares you for the Routing module. The Component Placement Setup lesson discusses the issues involved in preparing for component placement when creating geometries.



Objectives

This lesson provides an introduction to the features that control how components are placed on the circuit board. After reading the concepts and completing the lab exercise, you should be able to:

- Name the attributes that control placement and describe their usage.
- Name the properties that control placement and describe their usage.
- Describe the setup considerations for placement.
- Describe the elements that make up the layout environment.

Placement Controls

Attributes and properties provide control of where and how you place components on the board. You assign properties when you generate the schematic in Design Architect, although you can also add properties in PACKAGE. You add attributes when creating geometries in LIBRARIAN, although you can add additional attributes in LAYOUT.

The following sections discuss:

- The types of properties and attributes you may attach to symbols and geometries.
- The way in which properties and attributes affect component placement.

Component Controls for Placement

Attributes assigned to component geometries affect their placement. Some attributes are required for all components; others are optional and can be added at your discretion using the LIBRARIAN tool.

Component attributes that affect placement:

- **Component_placement_outline**—required attribute that defines the shape of the area covered by the placed component. Placement clearances are measured from the component placement outline.
- **Component_layout_type**—optional attribute used to define a surface mount component.
- **Component_layout_surface**—optional attribute used to allow placement of a component only on the back surface or both front and back surfaces of the board.
- **Component_diagonal_allowed**—optional attribute used to override board attribute Orthogonal_placement_only for diagonal placement of components.
- **Component_orthogonal_only**—optional attribute that limits specific component to orthogonal placement.
- **Component_height**—optional attribute that assigns a height to a geometry. Components using this geometry cannot be placed in a defined board placement region with height restrictions that would exclude them.
- **Component_outline_overhang**—optional attribute that allows a portion of a component outline to be placed outside the board placement outline as long as all component pins remain within the board placement outline.
- **Component_pins_moveable**—optional attribute that allows a component's pins to be moved interactively after placement in LAYOUT.

Property Controls for Placement

Properties attached to symbols in Design Architect affect placement. All placement related properties are optional.

Properties that affect placement:

- **Placement_region**—identifies a component as a member of a specific circuit group associated with a similarly named board placement region. Auto placement gives preference to components with this property when placing into the associated placement region.
- **Brd_loc**—specifies the location of a placed component. If forward annotated from the Schematic Editor, the presence of this property causes the owning component to be placed at the designated location when LAYOUT is invoked.
- **Swapping**—controls gate and pin swapping during placement improvement by overriding swap codes in the associated mapping file. Swapping property values are: Fixed (the gate is not swappable), Fixed_pins (the pins of the gate are not swappable), Fixed_all (the gates and the pins are not swappable), and Local (gates can swap only with other gates in the component).
- **Net_prio**—specifies how critical a net is. You can use this property to ensure that two components are placed closer together than they are placed otherwise. The Net_prio property value is an integer from 0 to 16. Sixteen is the highest priority.
- **Dec_cap**—associates the part number of a decoupling capacitor with a component during constructive placement.

Board Controls for Placement

Attributes control such aspects of placement as clearance between placed components, the placement grid, and keepout areas in which components cannot be placed.

You automatically add required board attributes when creating the board geometry in LIBRARIAN. Required board attributes can be changed in LAYOUT. You can add optional board attributes either when creating the board geometry or later in LAYOUT.

Board attributes that affect placement:

- **Board_placement_outline**—a required attribute that defines the area in which components can be placed.
- **Board_placement_keepout**—an optional attribute that defines an area of the board in which components cannot be placed.
- **Board_placement_grid**—a required attribute that defines the grid of points on which component origins are placed. This attribute can be overridden in LAYOUT.
- **Board_placement_clearance**—a required attribute that specifies the minimum allowable clearance between placed components. This attribute can be overridden in LAYOUT.
- **Board_placement_region**—an optional attribute that defines an area of the board with component height restrictions and/or associates the area with a circuit group whose members are components with the Placement_region property value of the circuit group name.

Placement Area and Keepouts

When you create the board geometry, you define a polygonal board placement outline limiting the area within which components can be placed. The area is the same for both sides of the board.

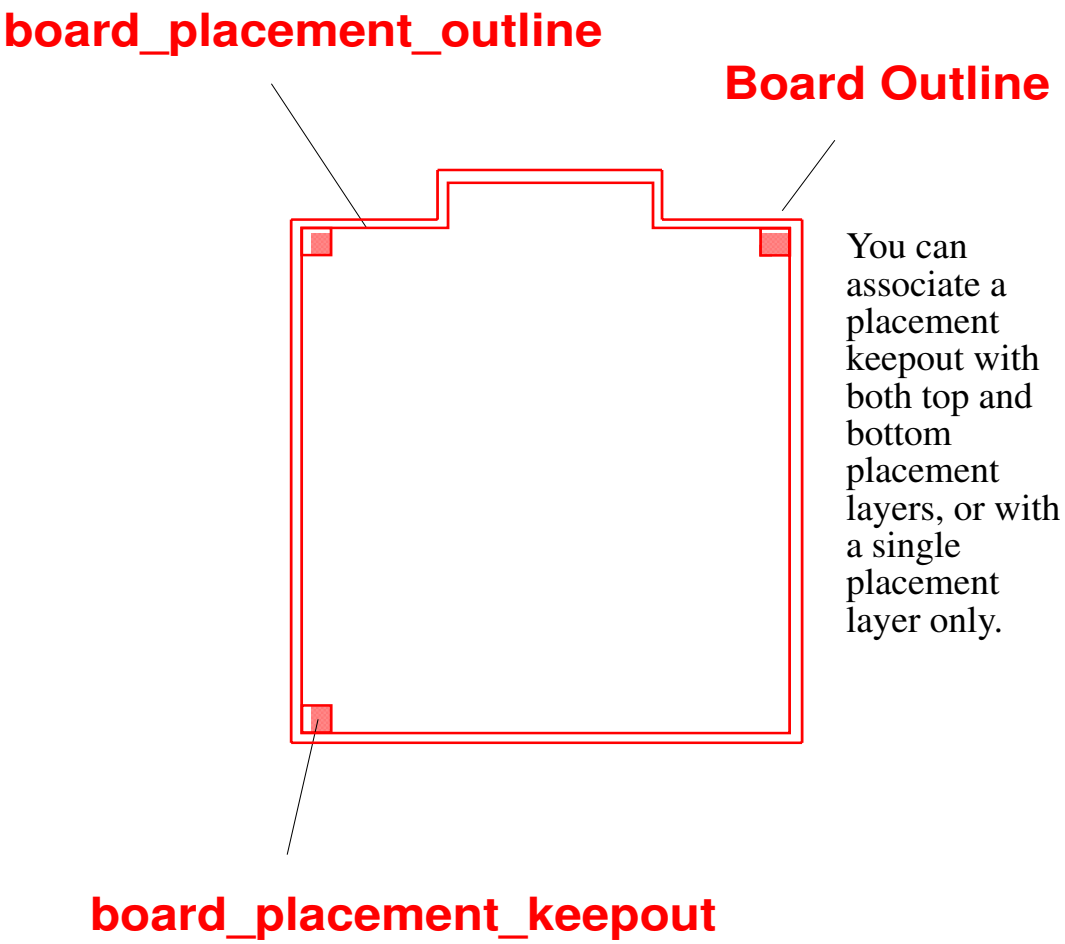


Figure 2-1. Placement Area and Keepouts

Placement keepouts are optional areas defined within the placement outline to mark locations in which you cannot place components. You can define a placement keepout with an optional attribute when you create the board geometry in LIBRARIAN. You can also define a placement keepout during a LAYOUT session. Placement keepouts can be moved and deleted in LAYOUT, as well as in LIBRARIAN. Whether created in LIBRARIAN or LAYOUT, placement keepouts always exist as attributes of the board geometry.

Placement Grid

The Board_placement_grid is a required attribute of the board geometry that defines an invisible grid of points within the placement area. Generally, this grid has the same origin as the displayed reference grid. You should define the reference grid in such a way that some, or all, of its points coincide with the placement grid.

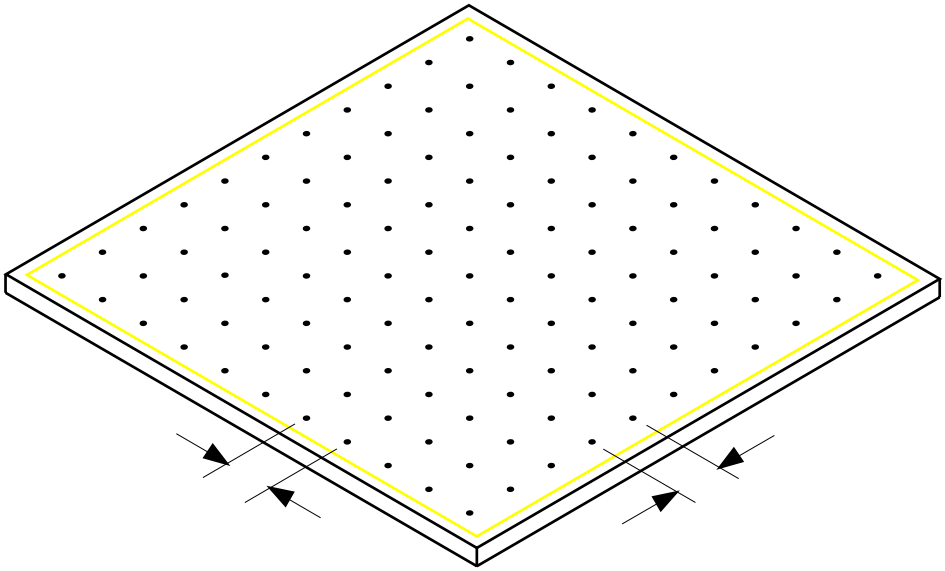
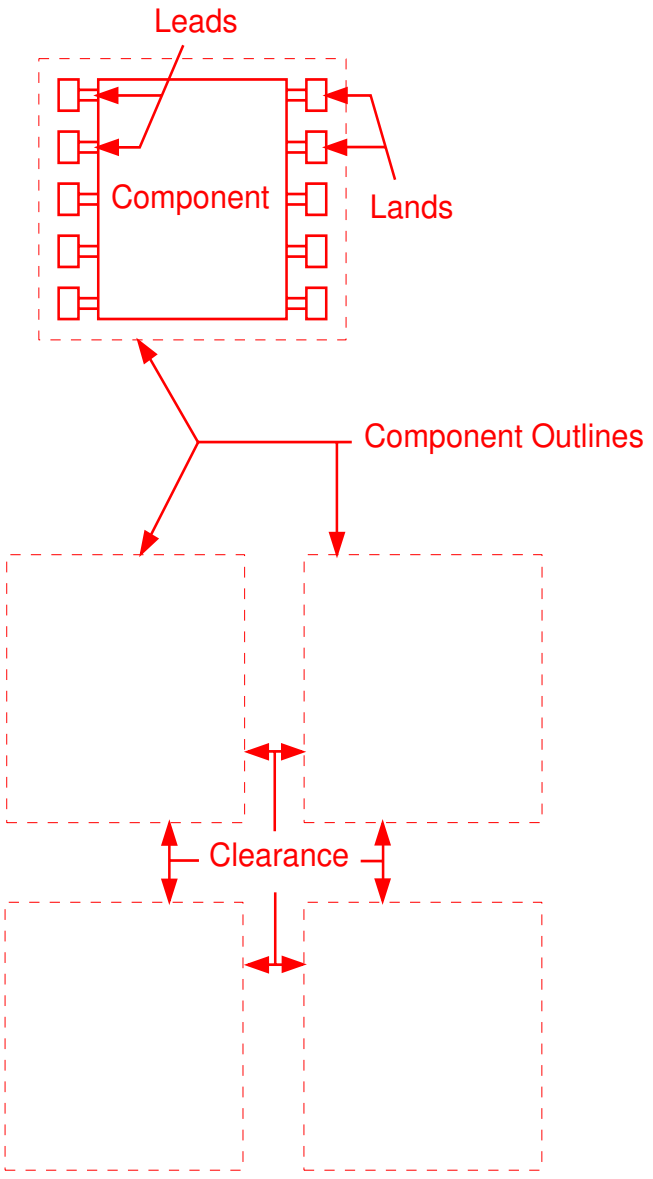


Figure 2-2. Placement Grid

During automatic placement each component is placed so that its origin coincides with a placement grid point. During interactive placement you may choose to snap component origins to the grid. Components with origins that are not on the placement grid are reported as placement warnings, and are not considered validly placed. For either automatic or interactive placement, you can choose to define a temporary placement grid that overrides the grid specified in the board attribute.

Placement Clearance

The required board attribute Board_placement_clearance determines the minimum distance allowed between placed components. Automatic checking uses this clearance value to prohibit placing or moving components too close together. During a LAYOUT session, you can define a temporary placement clearance to substitute for the permanent value set by the attribute.



Components can be placed closer together than the clearance when automatic checking is off. In this case, clearance violations are allowed, but the offending components are highlighted on the Errors layer.

Figure 2-3. Placement Clearance

Component Types

Component_type is a property allowing you to define a group of components that are related in some physical characteristic that could affect their component clearance requirements.

For example, a device that is hand-soldered due to thermal sensitivity requires a greater clearance to allow room for the hand soldering. Components placed by a pick-and-place machine might require a greater clearance than hand-placed components to allow room for the assembly equipment jaws. J-lead components might require greater clearance than components with gull-wing leads to allow for visual inspection and possible rework. These are a few reasons to group components into a type requiring a clearance that differs from the default board clearance.

For example, your default board clearance is .01, but you want components placed using a pick and place machine to have a clearance of .05. You attach the Component_type property to the components that will be placed with the pick and place machine and assign a value of PnP. In LAYOUT, you define the PnP component type and designate a clearance of .05 for components that are part of the PnP component type.

If you have any other component types, you can designate different clearance values between component types. For example, in addition to the PnP component type, you define another component type for components with J-leads. You call this new component type Jlead. You designate a clearance of .05 between the J-lead components and most other components on the board. However, because both the pick and place components and the J-lead components need additional clearance, if a J-lead component is placed next to a pick and place component, you designate a clearance of .075.

Table 2-1 illustrates the component type clearances in the example.

Table 2-1. Component Types and Clearances

	board default	PnP	Jlead
board default	.01	.05	.05
PnP	.05	.05	.075
Jlead	.05	.075	.05

You define a group by attaching the Component_type property either to the symbol in Design Architect or to the component in LAYOUT and assigning the same property value to the symbols and/or components you wish to group. The property value is any name you wish to designate.

In LAYOUT, you define the component type using menu item **Setup Placement > Component Types**. You then choose menu item **Setup Placement > Component Clearances** to designate placement clearance for the component type.

Component Height

You can define placement regions with minimum and maximum height restrictions. The placement routines in LAYOUT only allow components that meet these restrictions to be placed in these regions. This feature allows you to easily design the layout of the board according to the mechanical constraints of its surrounding environment.

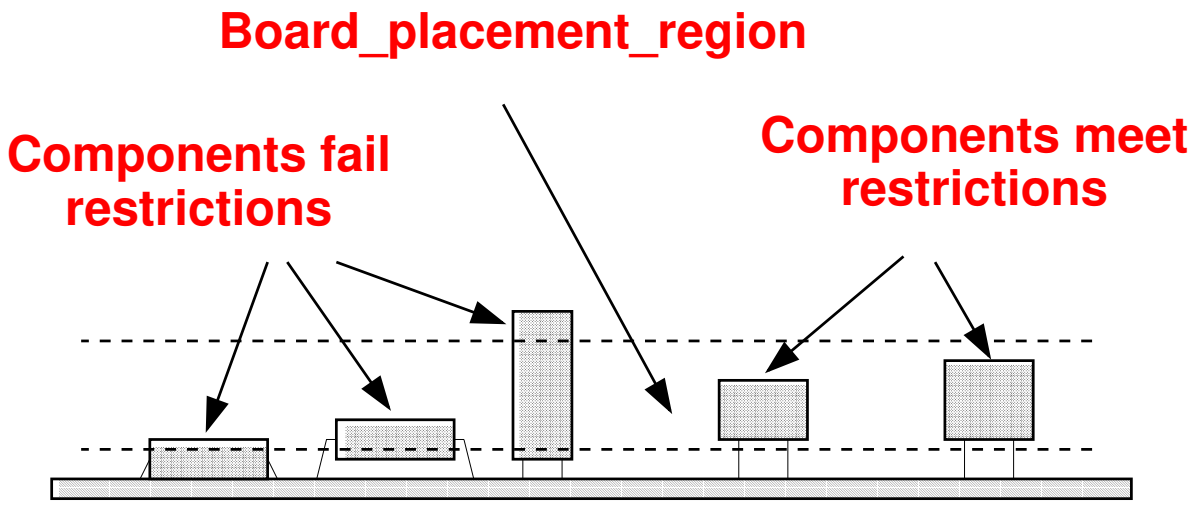


Figure 2-4. Placement Height Restrictions

Two attributes control component placement by height. The Component_height attribute assigns the height of a component. The Board_placement_region attribute defines placement regions with minimum and maximum height restrictions. During interactive and automatic placement, each placement region is considered a keepout area for all components with heights that violate the region's height restrictions. Because the region is considered a keepout rather than a keepin, components that meet the height restrictions do not have to be placed in the region.

Placement Regions

Sometimes a circuit designer prefers that certain logic functions in the schematic are packaged together and placed in the same region of the board. You can control this packaging and placement by assigning symbol instances to placement regions, creating a circuit group. The automatic placement routines in LAYOUT attempt to place circuit groups into the placement regions defined for them.

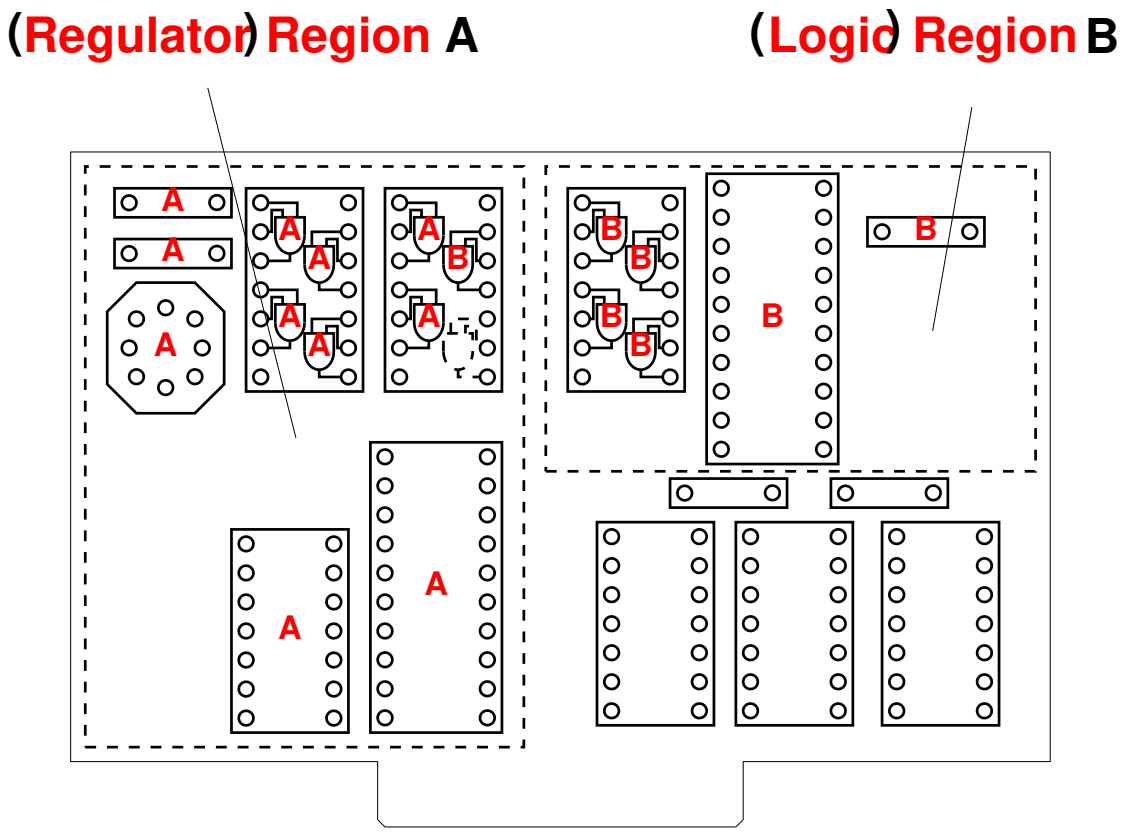


Figure 2-5. Placement Regions

You create circuit groups by attaching the symbol property Placement_region to symbols in Design Architect. The value of this property is a circuit group name. You can specify that PACKAGE assign symbols together if they are the same symbol type and contain the same circuit group name as the Placement_region property value.

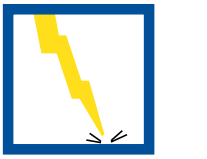
You define a placement region using the board attribute, Board_placement_region. Define the Board_placement_region attribute using the circuit group name that matches the value of the Placement_region property.

During automatic placement, LAYOUT places components in a circuit group into the matching placement region, if possible. Otherwise, the standard constructive placement rules apply. Automatic swapping of gates among components belonging to different circuit groups is not allowed; however, you can swap these gates interactively.

In Figure 2-5, note that one B gate is placed in Region A. Gate B is in Region A because, by default, PACKAGE minimizes the number of packages used, even if that requires putting gates from different circuit groups into the same component. The placement region assigned to each component is biased toward (but not solely determined by) the dominant circuit group assigned within the component.

Be aware that only the autoplacer respects placement regions. Using interactive placement, you can place any component in any region of the board, as long as you do not violate placement clearances.

Placement Regions for Split Planes

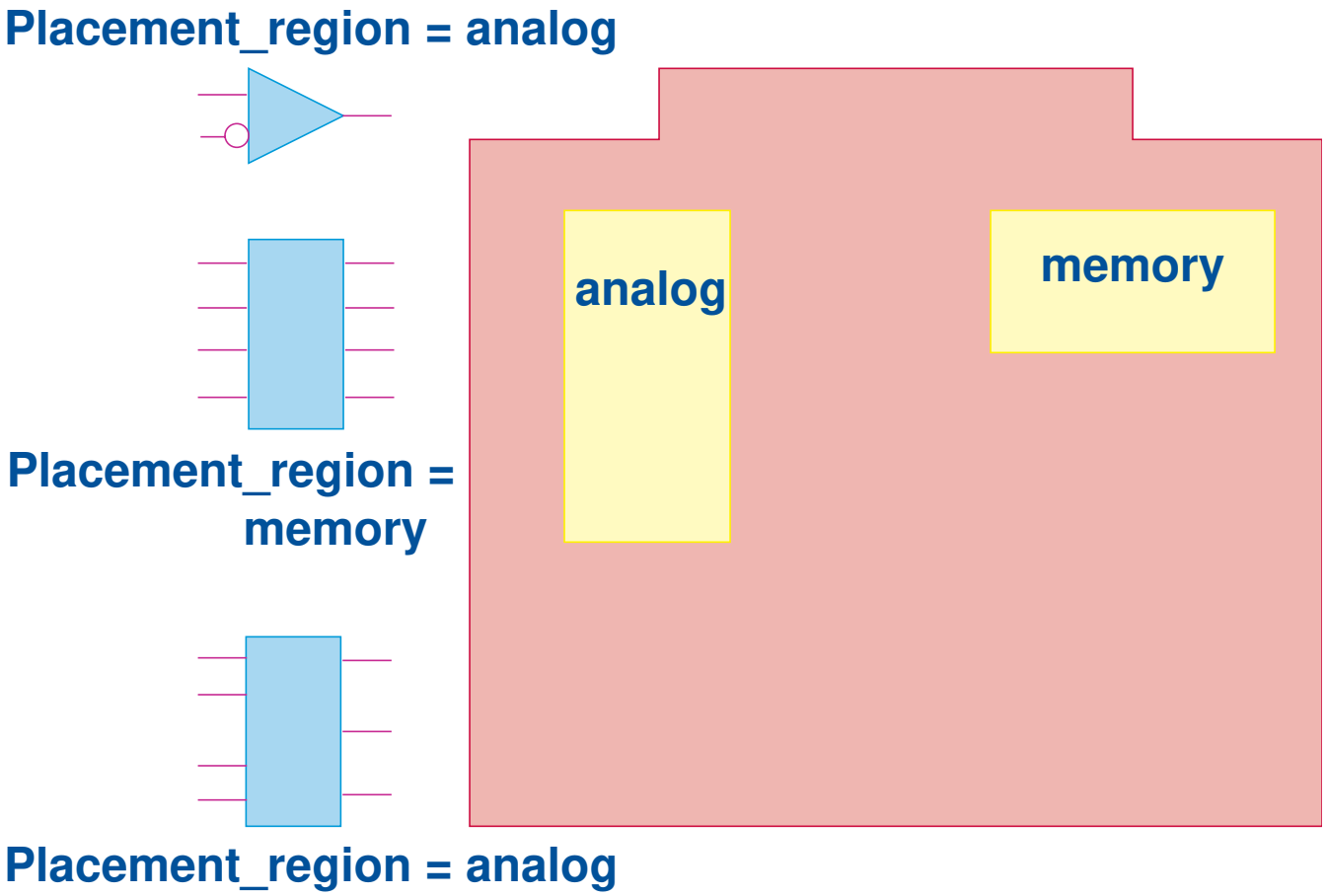


Split Power Plane

One application of placement regions is to facilitate creating split power planes. You want to place components together that connect to the alternate power. Placing these components in close proximity and, as much as possible, separate from components connecting to the main power plane makes it easier to create the split plane.

To prepare for split power planes you:

- Attach the Placement_region property to symbols in Design Architect giving the symbols a value that you also use as the placement region name.
- Define a placement region on the board giving the placement region the same name you used for the value of the Placement_region property in Design Architect.



Placement Issues

Special features on a circuit board require consideration early in the design process.

- Split power planes (power nets).

  Split power planes and area fills require careful analysis of component placement and grouping.

- Area fills.

- Artwork order.

  Artwork order geometry discussed in the Creating Manufacturing Data module requires setup to handle these requirements.

- Etched text.

- Reference designators.

  Use of etched text and reference designators also requires careful placement planning.

Setup Controls for Placement

In LAYOUT you can override the required Board_placement_clearance and Board_placement_grid attributes of the board geometry to change how components are placed.

- Override Placement Clearance

  You can set a new placement clearance value.

- Change Placement Grid

  You can define a different placement grid.

- Snap to Placement Grid

  You can control whether component origins are placed on the placement grid during interactive placement.

Any such changes you make in LAYOUT are saved with the design environment.

LAYOUT Settings

You can set up display and edit settings and save the settings when you close a LAYOUT session. These settings make up the LAYOUT environment. Each time you return to LAYOUT, the saved settings are re-established. These environment settings include:

- View layers
- View objects
- View styles
- Display attributes for layers and objects
- View component labels
- Set up edit layer
- Automatic checking

Lab Exercise

In the following lab exercise you set a number of environmental settings in LAYOUT. These settings customize view and layout parameters for use in all of your placement and routing sessions.

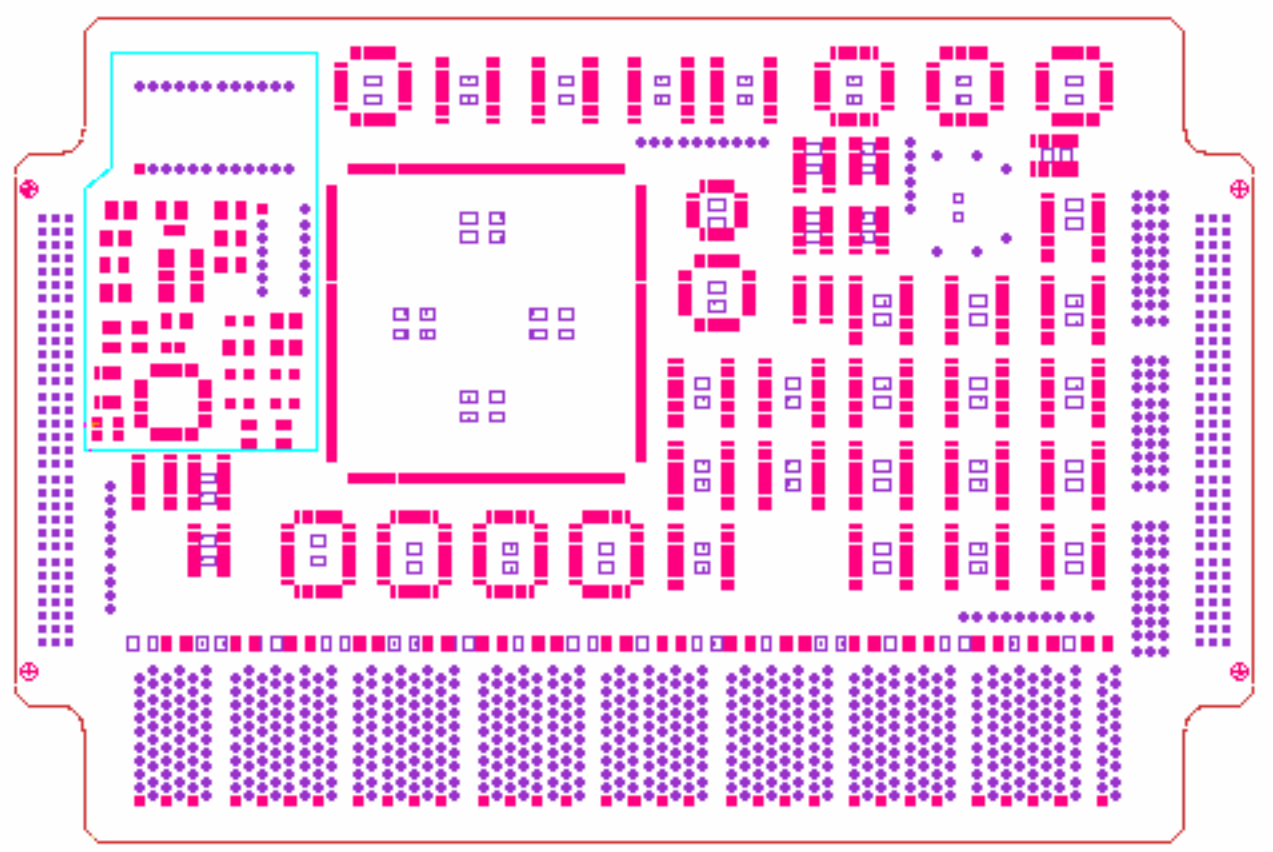
Upon completion of this lab exercise you should be able to:

- Set up the environment to:
  - View polygons as filled and text as centerline.
  - Include placement and router keepout in view layers.
  - View references as component labels.
  - View guides as nets from selected components.
- Set up placement regions.
- Set a placement grid.

Turn to Module 5—Lab 2: "Component Placement Setup".

Lab 2
Component Placement Setup

The Placement and Routing labs give you experience with the major features of the LAYOUT tool. The Placement module acquaints you with the interactive and automatic placement features and prepares you for the Routing module. The Component Placement Setup lab shows you how to set up parameters for component placement.



Introduction

In this lab exercise, you preset a number of environmental settings in LAYOUT. These settings customize view and layout parameters for use in all of your placement and routing sessions.

Upon completion of this lab exercise you should be able to:

- Set up the environment to:
  - View polygons as filled and text as centerlined.
  - Include placement and router keepout in view layers.
  - View references as component labels.
  - View guides as nets from selected components.
- Set up placement regions.
- Set a placement grid.

Procedure

You use LAYOUT to preset the environmental settings for your session.

Preparation for Lab

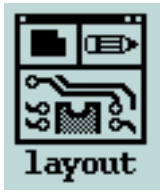
To set up for this lab you need to invoke the LAYOUT tool on your design.

1. If you or your instructor have not already done so, complete the Installation Procedure in the About This Training section of this manual.

2. Invoke the Design Manager by entering the following in a shell:

   ```
   $MGC_HOME/bin/dmgr
   ```

3. Using the Design Manager, change your current directory to the board_new directory by clicking on the four-way icon in the navigator window. In the **Change directory to** dialog box, enter the pathname: your_path/**training/board_new/mod5** and press the Return key.



4. Find the LAYOUT icon in the Tools window. Invoke LAYOUT by placing the cursor on the LAYOUT icon and double clicking the Select mouse button.

   The INVOKING LAYOUT: Select a Design dialog box displays.

5. Select the **sig_az** design and **OK** the dialog box.

   If an Invocation Switch dialog box appears, select the **OK** button without selecting the switch.

   A Report-Startup message appears in the middle of the LAYOUT Session window. This is a list of notes concerning the files used to invoke the LAYOUT tool. After reading the report notes, **Close** the report window, and then maximize the size of the LAYOUT session window to fill the display.

Setting up Placement

The design parameters you set, such as View Style, Edit Layer, and View Layers are used each time you use a Board Station tool. To save setup time, these parameter settings are saved each time you **Close** a design session. In this lab you review, change, and save some of the placement design parameters.

Setting and Understanding the Display

1. The icon menu at the upper-right corner of the Edit window allows quick access to viewing functions.

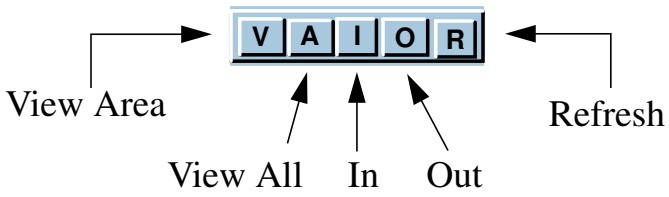
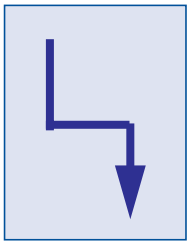


Figure 2-6. Icon View Menu

2. Click the Select (left, by default) mouse button on the different buttons to try out the functions.

Controlling the Display



In this section, you set various controls to determine whether and how you view the layers and objects that make up your design.

1. Choose the **View > Display Controls** menu item. Or, use the Display Controls stroke to display the dialog box.

   Some of the following settings may already be in place from the previous lab. If so, keep the settings and continue with the next step.

2. In the Objects section of the dialog box, select to view only the following objects.

   Drill_holes

   Errors

   Geom Attributes

   Pads

   Text

3. In the Layers section of the dialog box, click on the **View Layers...** button.

   The View Layers dialog box displays. A two-column list of the layer names are listed in the dialog box, with each name appearing in the color in which graphics on that layer are created. Next to some of the layer names are the letters *s* and *v*. If an *s* is visible next to a layer name, the graphics on that layer are selectable. If a *v* is visible next to a layer name, that layer is visible.

A *V* next to each name indicates visibility.

Next you set the view layers for the placement and routing keepout layers. These layers help you see legal component placement areas. Select to view only the following layers:

Board_outline

Place, Placement_region_1, Place_1, Place_2

Signal, Signal_1, Signal_2, Signal_3, Signal_4

4. **OK** the View Layers dialog box.

   In the previous lab exercise, you set the board background to a color other than black. Now, reset the background to black.

5. Click on the **Display Attributes...** button.

6. Select the **Objects and Attributes** button.

   The list of objects and attributes displays along with the colors and patterns currently assigned to each.

7. Set the **Background** back to black and **OK** the Set Display Attributes dialog box.

8. In the Other Options section, select **Component Label...**

9. In the View Component Labels dialog box, choose Label Type: **Reference**, and then **OK** the dialog box.

   Even though you do not place components in this lab, turning the reference designators on now makes it easier to locate specific components when you do begin placement.

10. In the View Style section of the dialog box, select to view the Polygon/Trace objects as **Centerline** and Text objects as **Outline**.

11. **OK** the Display Controls dialog box.

Set the Select Filter to Select Components.

1. Choose the **Setup > Select Filter...** menu item.

2. In the Setup Select Filter dialog box, select only the **Components** option. Ensure no other options are highlighted. **OK** the dialog box.

   Now, only components can be selected. Limiting selection to components only during placement makes moving components easier, because you do not have to worry about accidentally selecting other geometries.

Set the connection display for rubber banding

When you place components, the View Connections option displays the guides from any selected component. If the Guides layer is visible, the guides for the selected component highlight in a different color than the guides of non-selected components. If the Guides layer is not visible, the guides for the selected component highlight and are the only visible guides.

1. Choose the **Setup Placement > Interactive Placement > View Connections...** menu item.

   In the View Connections dialog box, you can choose to display:

   Connections--guides that directly connect to pins of the selected component.

   Nets--all guides from the selected component.

   None--no guides display.

   You can also choose to display:

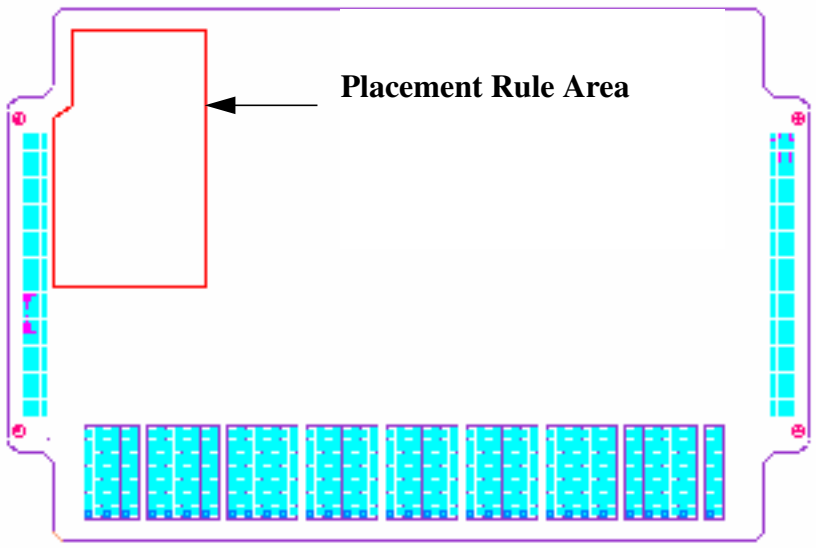
   Connection Dynamics--the rubber banding effect of guides moving with the component.

2. Choose **Nets** and **Connection Dynamics**. **OK** the dialog box.

Defining a Placement Rule Area

Define a Placement Rule Area for the analog components. The autoplacement routines place components with the specified Placement_region property into the matching Placement Rule Area. Even though a Placement Rule Area exists, you can still interactively place any component anywhere on the board. Only the autoplacement routines respect Placement Rule Areas.

Examine the Placement Rule Area



1. View an area around the Placement Rule Area in the upper-left corner of the board.

2. Click the Select mouse button on the boundary of the placement rule area.

   Can you select it? Why not? The Select Filter is set to allow selection of Components only.

3. Choose the **Setup > Select Filter...** menu item.

   In the Setup Select Filter dialog box, click on **Geom Attributes** so that option is selected. Click on **Components** so that option is no longer selected. Ensure only **Geom Attributes** can be selected. **OK** the dialog box.

4. Again, click the Select mouse button on the boundary of the placement rule area.

5. Choose menu item **Report > Selected**. Examine the report. Note that the Placement Rule Area is an attribute. When you finish, close the report and delete the attribute.

Create a New Placement Rule Area

1. Choose menu item **Setup > Grid...**. Enter an X-increment value of **.05** and **OK** the dialog box.

2. Choose the **[Top Menu] Placement > Add Placement Area > Add Placement Rule Area...** menu item. In the Add Placement Rule Area prompt bar, enter the following information and **OK** the dialog box.

   Side: **Top**
   Fill Pattern: **4**
   Maximum Height: **[Leave this box blank]**
   Minimum Height: **0.0**
   Circuit Group: **analog**

   A prompt bar displays and the cursor changes to the crosshair location cursor.

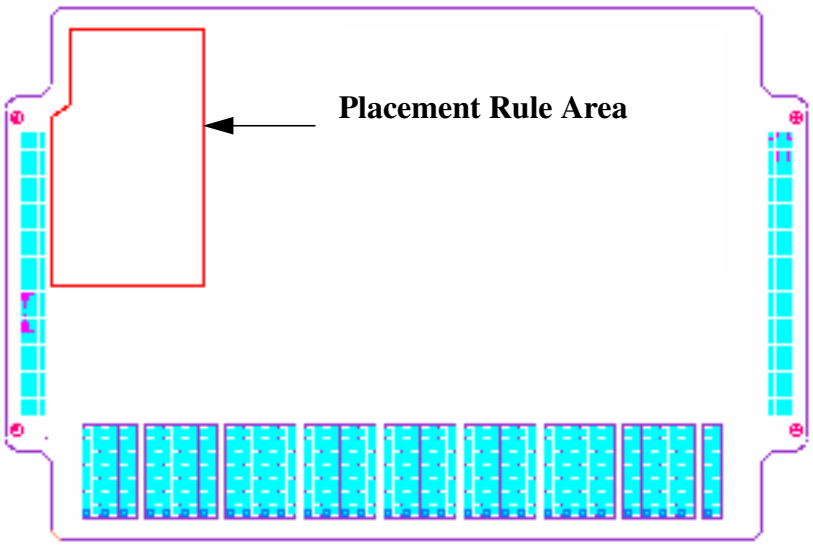
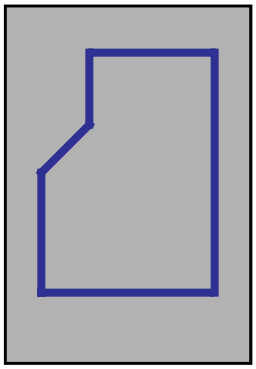


Figure 2-7. Analog Placement Region



3. Redefine a Placement Rule Area in the upper-left corner of the board by clicking the Select mouse button at the absolute coordinates of the polygon shown in Figure 2-7. The coordinates are:

1.7 5.75
1.7 2.85
0.0 2.85
0.0 4.75
0.2 4.9
0.2 5.75
1.7 5.75

Alternatively, you can enter absolute coordinates by typing in the Edit window. Type **coo** *for coordinate. Then type the coordinate location and hit the carriage return. For example:* **coo 1.7 5.75**

4. **OK** the prompt bar to complete the polygon.

This design has an analog section on it. The components in the analog section each have a property name *Placement_region* with a value of *analog*. You added this property to some of the analog components in the packaging lab. Components with this property name/value combination can only be autoplaced in the placement area you just defined.

Setting the Placement Grid

When you created the board outline, you defined the placement grid as 0.10. If the design criteria change, you can change the placement grid without going back to LIBRARIAN.

1. Choose the **Setup Placement > Placement Grid:** menu item. In the prompt bar, specify a Placement Grid of **0.05**.

2. **OK** the prompt bar.

Reset the Select Filter to Select Components

1. Choose the **Setup > Select Filter...** menu item.

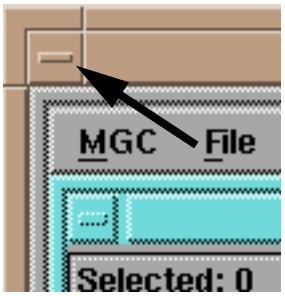
2. In the Setup Select Filter dialog box, select only the **Components** option. Unselect the **Geom Attributes** option. Ensure only the Components option is highlighted. **OK** the dialog box.

Saving Design Data and Leaving LAYOUT

Now that you have set the placement parameters, you can save the data to make it available for the rest of the design process. You save the placement parameters to the design. The next time you invoke the LAYOUT tool on this design, LAYOUT reads those parameters and puts them into effect for the session.

1. Choose the **File > Save > Design All** menu item.

   After you are sure the design data has been saved, you have the option of continuing in LAYOUT or closing the session. Using the **File > Save > Design All** menu item is the safest way to save design data, because if something is not saved the way you like, the session is still open for you to save again.



2. Close the session by choosing **Close** from the Window menu (upper-left most window icon).

   When you close the session, the **Save Changes to Design** dialog box appears. If you have already saved your data and back annotated to the PCB design viewpoint, it is not necessary to save the data again. However, it is good design practice to save data and back annotate as you leave LAYOUT. Saving data and back annotating as you close the LAYOUT tool ensures that the layout and the PCB design viewpoint stay in sync.

3. Choose **Yes** and the dialog box progresses to ask if you want to **Back Annotate the PCB Design Viewpoint**.

4. Select the button indicating to **Back Annotate the PCB Design Viewpoint**.

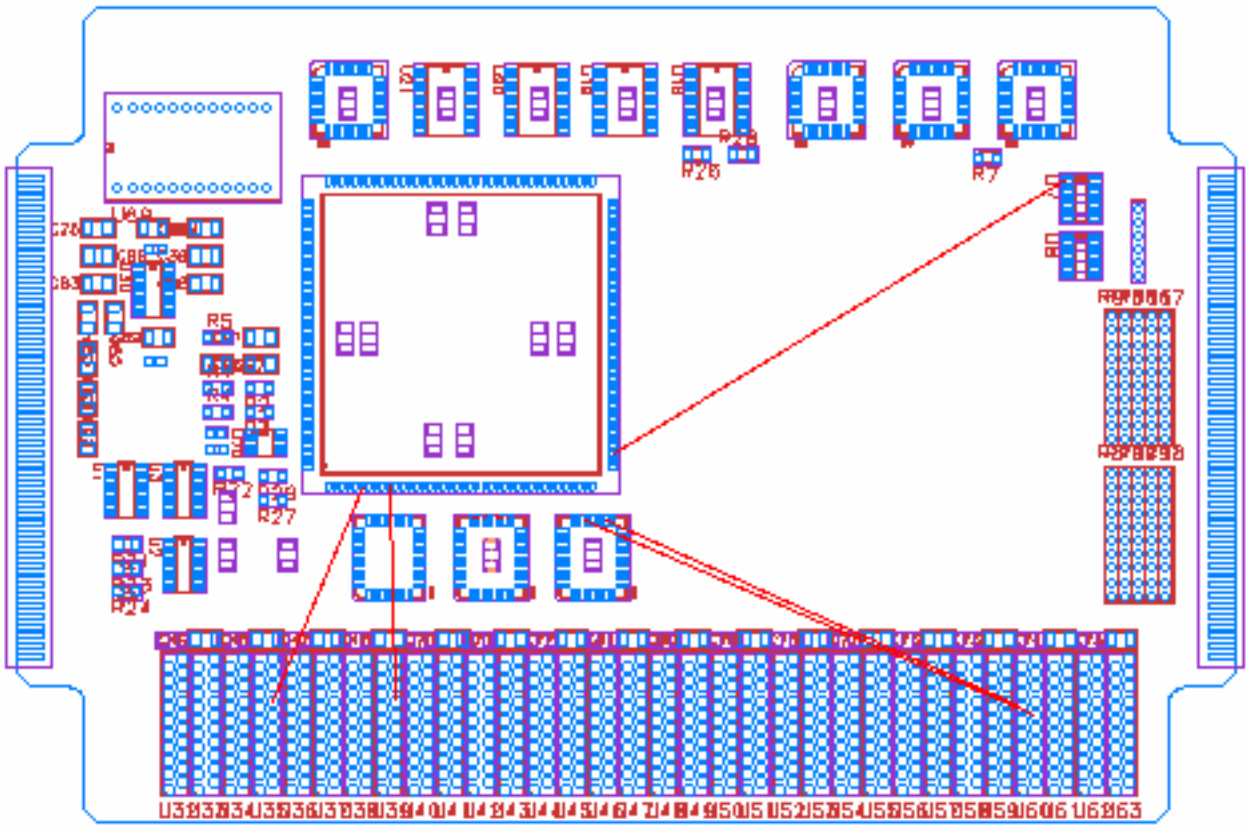
5. **OK** the dialog box.

6. Close the Design Manager.

Congratulations! You have completed the "Placement Setup" lab exercise. Continue with Lesson 3: "Interactive Placement".

Lesson 3
Interactive Component Placement

This lesson in the Placement and Routing modules also provides an introduction to the major features of the LAYOUT tool. The Placing Components on a Circuit Board module acquaints you with the interactive and automatic placement features and prepares you for the next module, Routing. The Interactive Component Placement lesson discusses placing components on the board interactively.



Objectives

This lesson provides an introduction to the interactive placement features of the LAYOUT tool. After completing the concepts and lab exercise in this module, you are able to understand how to:

- Interactively place components on the board.
- Adjust the placement of components on the board.

Interactive Placement - By Reference

You can interactively place any component, unplaced or mapped off the board, by entering its reference designator into the Place Component by Reference dialog box. The function selects the specified component and allows you to drag it to the desired location on the board.

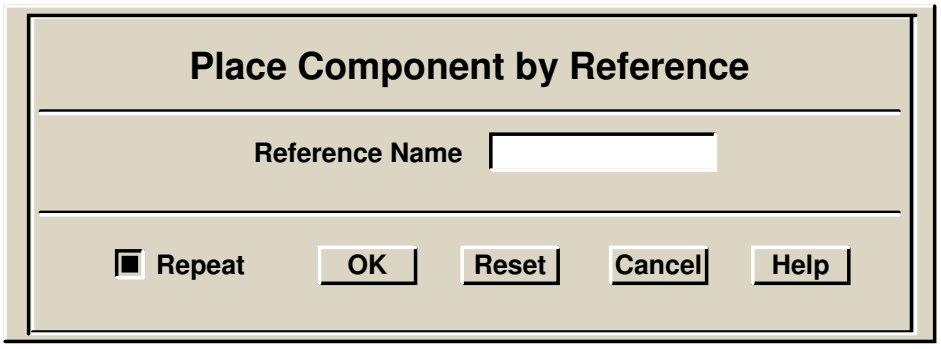


Figure 3-1. Selecting for Placement by Reference Designator

To use the Place Component by Reference function, place the cursor in a board geometry edit window, and choose the **Placement > Extended Menu > Place Components > Place Component by Reference...** menu item from the popup menu.

Interactive Placement - By Connectivity

The place-by-connectivity mode automatically selects the unplaced component, or component mapped off the board, with the most connections to previously placed components, and initiates a move operation. You place the component on the board by dragging the component to a placement location and then pressing the Select mouse button.

Figure 3-2 shows connectivity between the component about to be placed, which is to the lower-left of the board, and component already placed on the board.

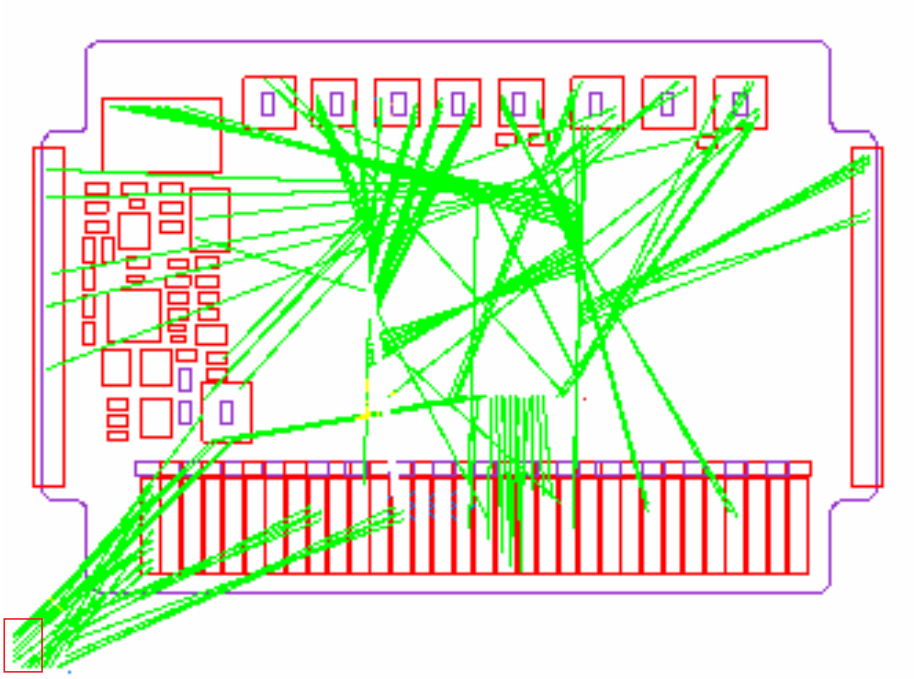


Figure 3-2. Components are Places by Connectivity

As soon as you finish placing a component, another is selected automatically and you repeat the process. This continues until there are no more components remaining to be placed, or until you cancel the function. Using this method, you can place components, one at a time, in decreasing order of connectivity.

Interactive Placement - From Schematic

You can interactively place components by selecting gates shown on a schematic and dragging the cursor to the board layout. You display a sheet of the schematic by choosing the **View > Schematic Sheet > Schematic Sheet:** menu item.

Filling out the prompt bar allows you to navigate to the sheet you want to view. For example:

Hierarchy: /DATA_IO/HI_SPEED_CHANNEL
Sheet: sheet1

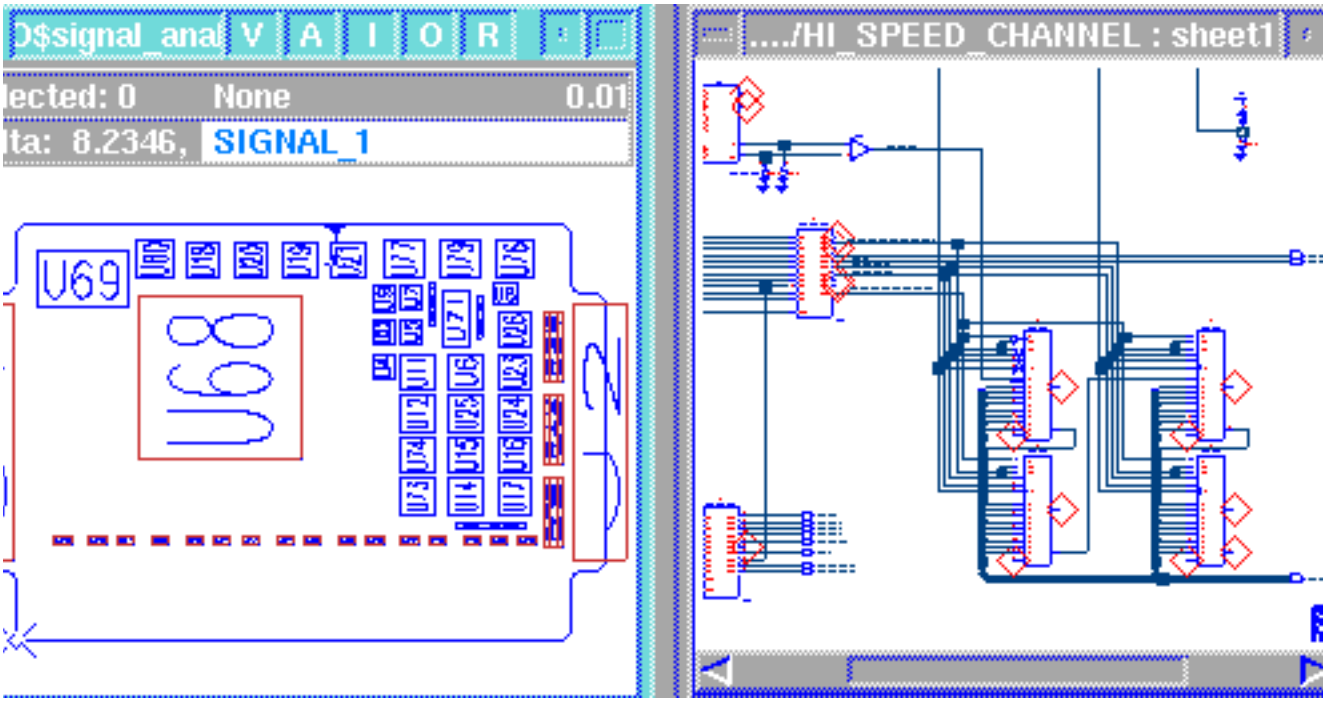


Figure 3-3. Placing from the Schematic

To view other areas of the schematic hierarchy, enter the path to the specific instance you wish to view. To view other sheets of the schematic, enter the appropriate sheet name.

The following steps outline how you select a gate on the schematic sheet and place the component on the board.

1. To select components for placement, choose the **Place Component** menu item from the Schematic window popup menu.

   A prompt bar appears and the cursor changes to a crosshair for selection.

2. Place the cursor on the gate of the component you wish to place and click the Select mouse button.

3. To place the component, move the cursor to the Edit window. As the cursor enters the Edit window an outline of the selected component follows the cursor. Position the component at the desired location and click the Select mouse button.

   The component is placed on the board.

When you view the schematic from LAYOUT, remember that you are viewing the sheets through the design viewpoint. The schematic is not editable from the LAYOUT tool.

Moving Components

You can move a single component or a group of components. Preselection is not required. If no components are selected, LAYOUT prompts to select components within an area before proceeding with the move.

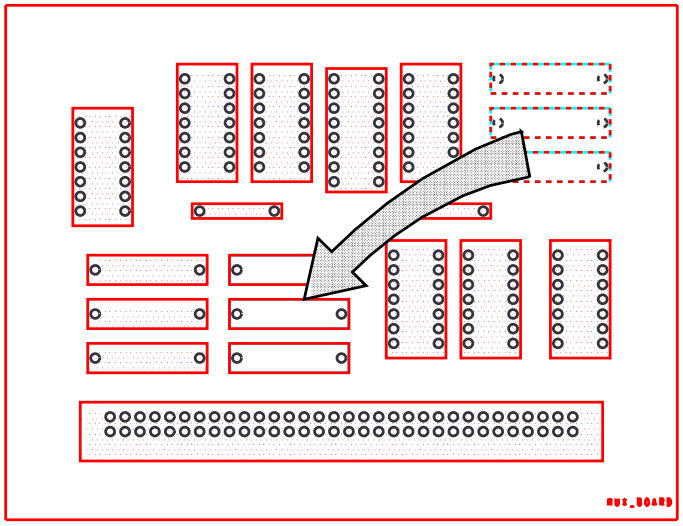


Figure 3-4. Selecting a Component to Move

Choose the **Placement > Move** menu item from the Edit Window popup menu.

Figure 3-5. Move Prompt Bar

If LAYOUT prompts you to select components, but no components highlight when you define the selection area, press the Options button on the Select Area prompt bar. In the Select Area (Options) dialog box that displays, press the Components button to temporarily set the selection filter to components.

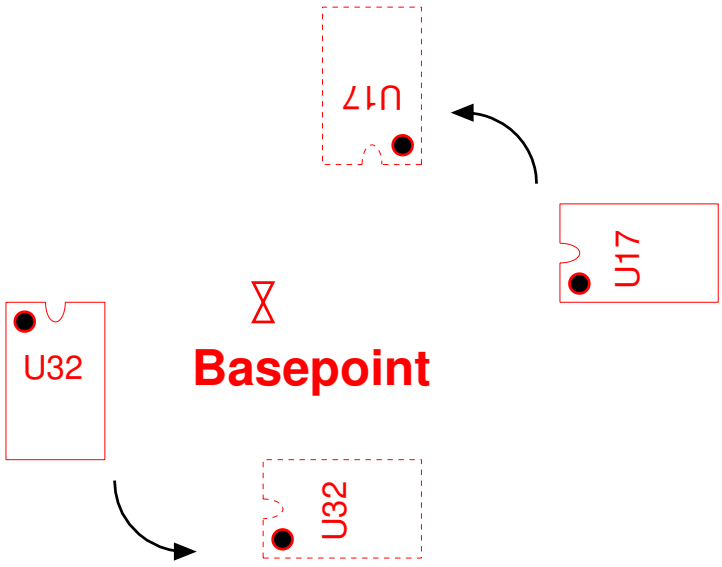
Rotating Components

You can rotate a single component or a group of components. Rotating revolves the component around the basepoint of the component. Preselection is not required. If no components are selected, LAYOUT prompts you to select components within an area before proceeding.

Choose the **Placement > Rotate:** menu item from the Edit Window popup menu.

Figure 3-6. Rotate Prompt Bar

If LAYOUT prompts you to select components, but no components highlight when you define the selection area, press the Options button on the Select Area prompt bar. In the Select Area (Options) dialog box that displays, press the Components button to temporarily set the selection filter to components.



You can rotate the selected component(s) by dragging to the desired angle, or by typing the angle in the prompt bar. The angle can be relative to the current orientation of the selected component(s), or it can be an absolute angle.

Figure 3-7. Rotating Components

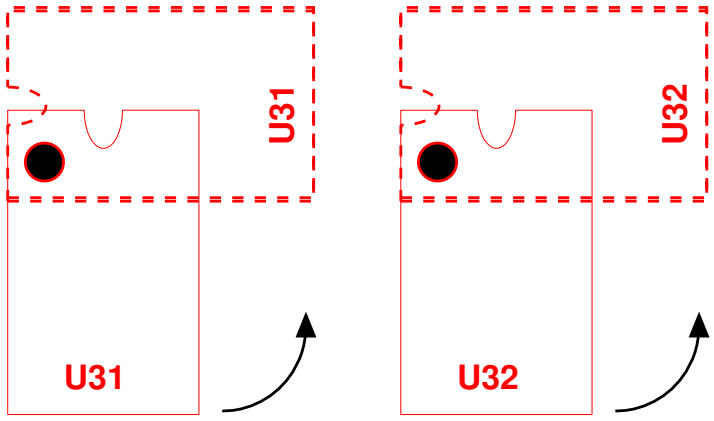
Pivoting Components

You can pivot a single component or a group of components. Pivoting revolves a component around the origin or center of the component. Preselection is not required. If no components are selected, LAYOUT prompts you to select components within an area before proceeding.

Choose the **Placement > Pivot** menu item from the edit window popup menu.

Figure 3-8. Pivot Prompt Bar

If LAYOUT prompts you to select components, but no components highlight when you define the selection area, press the Options button on the Select Area prompt bar. In the Select Area (Options) dialog box that displays, press the Components button to temporarily set the selection filter to components.



You can pivot the selected component(s) by dragging to the desired angle, or by typing the angle in the prompt bar. The angle can be relative to the current orientation of the selected component(s), or it can be an absolute angle. You can choose to pivot the component around its origin, or around its center. If more than one component is selected, each pivots around its own origin or center

Pivoting 90 Degrees Around the Origin

Figure 3-9. Pivoting Components

Flipping Components

Flipping a component switches a component placed on the top, or front, of the board to the bottom, or back, of the board or vice versa. By default, a component is flipped around a vertical axis through its center. Preselection is not required. If no components are selected, LAYOUT prompts you to select components within an area before proceeding.

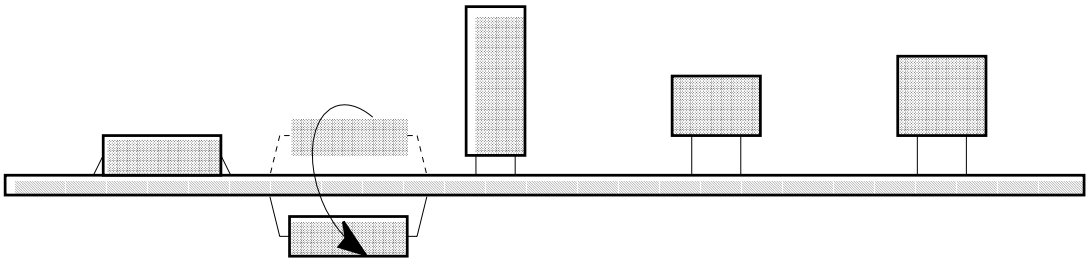


Figure 3-10. Flipping Components

Choose the **Placement > Flip** menu item from the Edit Window popup menu.

If LAYOUT prompts you to select components, but no components highlight when you define the selection area, press the Options button on the Select Area prompt bar. In the Select Area (Options) dialog box that displays, press the Components button to temporarily set the selection filter to components.

Protecting Components

You can protect individual components, component groups, or all components in a design. Protected components cannot be moved by subsequent placement operations. This includes placing, moving, pivoting, rotating, flipping, aligning, swapping, or automatic mapping.

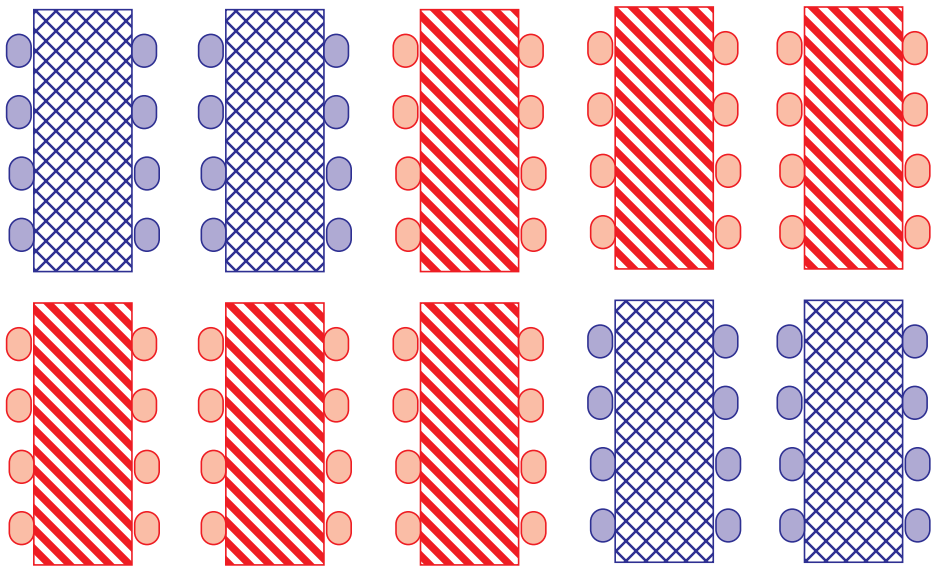


Figure 3-11. Protected Components Highlight

You can protect unplaced components, but protected components cannot be placed until they have been unprotected. By default, protected components highlight in red.

For more information on protecting components, refer to the section "Protecting Components" in the *LAYOUT User's Manual*.

Fixing Component Placement

A placed component can be fixed in its location. When a component is fixed, it is treated similarly to a protected component, but it also has the additional characteristic of being logically associated with the board geometry. This logical association is accomplished through the Fixed_component_location attribute assigned to the board geometry when the component is fixed.

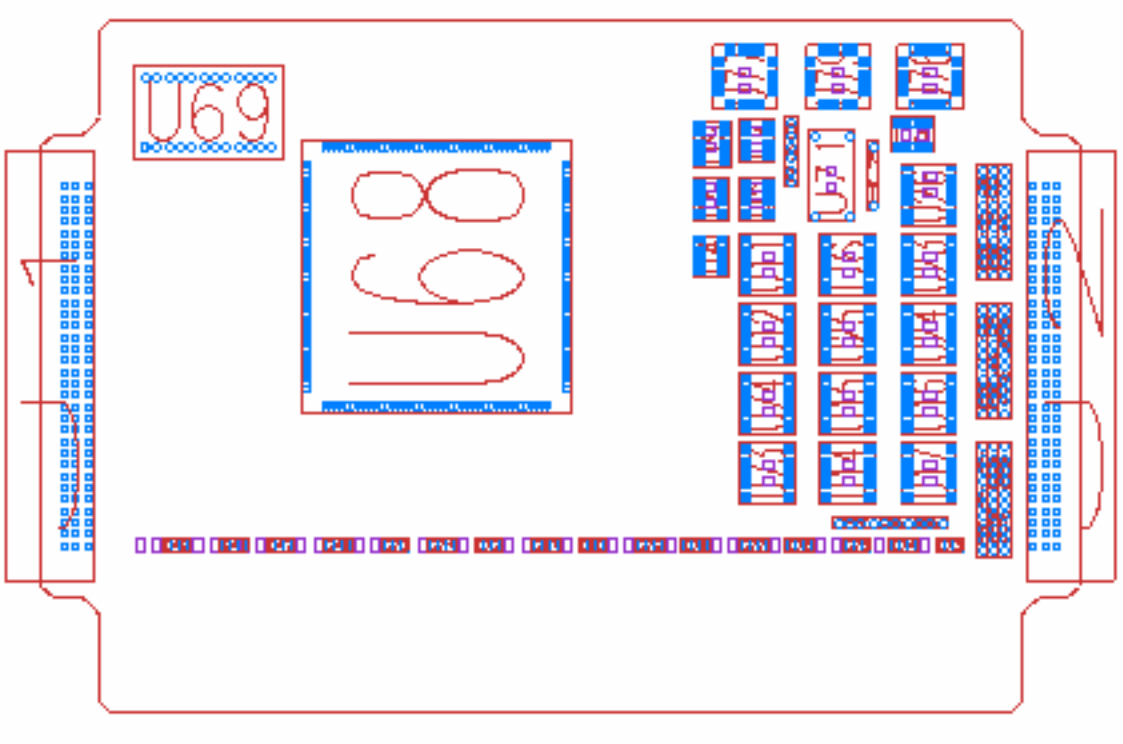


Figure 3-12. Connectors are Commonly Fixed in Position

There are two methods for fixing components. You can either:

- Fix selected components.
- Specify the reference designators of all components you want fixed.

Connectors are a common component to fix on the board.

Mapping Components

Component mapping is a way to place components on or off the board without regard to connectivity. You can use it to place a group of selected components off the board to later move them individually onto the board. You can also use mapping to generate an initial placement, as components are mapped onto the board in no particular order. You then use interactive features such as move, rotate, and flip to achieve the desired placement.

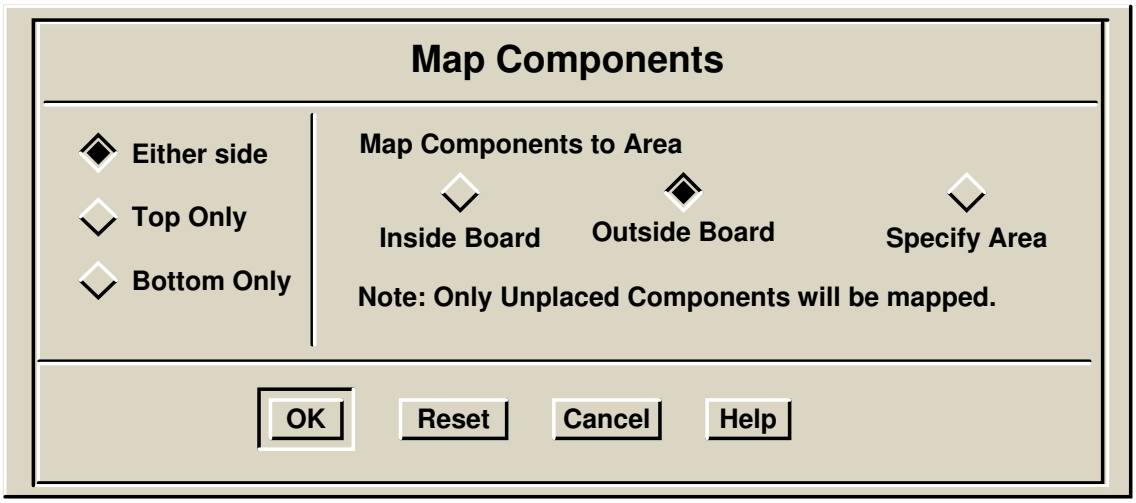


Figure 3-13. Map Components Dialog Box

Use the **Auto Placement > Map Components...** menu item from the edit window popup menu. Mapping options in the Map Components dialog box are as follows:

- **Either Side/Top Only/Bottom Only**—specify the side of the board onto which to map components.
- **Inside Board**—maps components onto the board using the entire placement area of the board. You have the choice of mapping all or only unplaced components.
- **Outside Board**—maps components into an area outside the board placement outline.
- **Specify Area**—allows you to define a map area inside or outside the board. You have the choice of mapping all or only unplaced components.

Lab Exercise

In this lab exercise you examine the process of interactive component placement. You select components for placement based on the values of properties such as reference designators. You also examine the basic techniques for moving, rotating, and pivoting components.

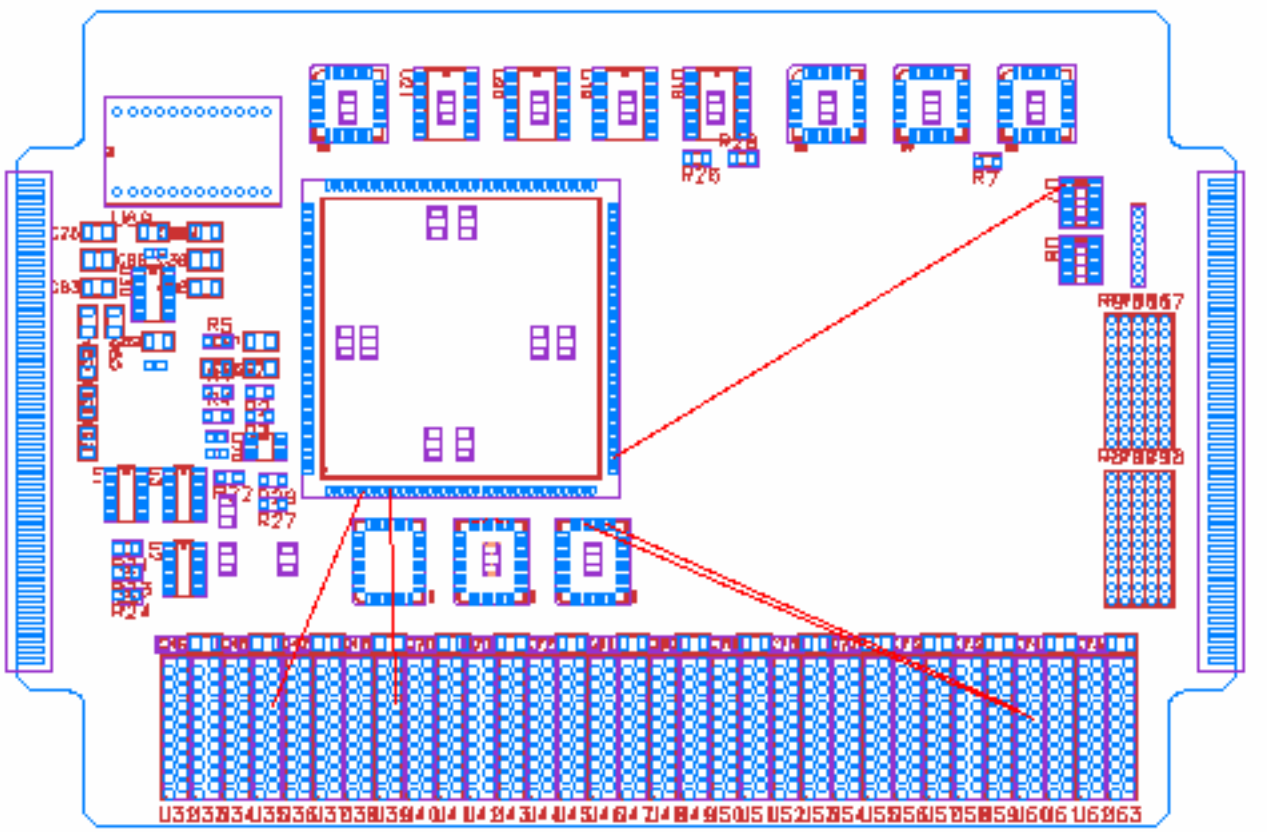
Upon completion of this lab exercise you are able to:

- Place components by reference.
- Place components by connectivity.
- Move, rotate, and pivot components during and after placement.
- Map remaining components off the board.
- Complete a placement interactively.

Turn to Module 5—Lab 3: "Interactive Component Placement".

Lab 3
Interactive Placement

The Placement and Routing labs give you experience with the major features of the LAYOUT tool. The Placing Components on a Circuit Board module acquaints you with the interactive and automatic placement features and prepares you for the Routing module. In this lab you gain experience placing components on the board interactively.



Introduction

In this lab exercise, you examine the process of interactive component placement. You select components for placement based on the values of properties such as reference designators. You also examine the basic techniques for moving, rotating, and pivoting components.

Upon completion of this lab exercise, you are able to:

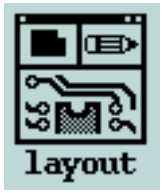
- Place components by reference.
- Place components from the schematic.
- Place components by connectivity.
- Move, rotate, and pivot components during and after placement.
- Map remaining components off the board.
- Complete a placement interactively.

Procedure

You use LAYOUT to interactively place components on the board.

Preparation for Lab

You use several different methods to place the components. The board must be only partially placed, including the connectors that were placed and fixed using the LIBRARIAN tool.

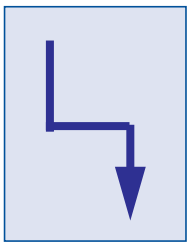


1. Invoke the Design Manager.
2. From the tools window of the Design Manager, invoke LAYOUT on the **sig_az** design.

   If an Invocation Switch dialog box appears, select the **OK** button without selecting the switch.

   A Report-Startup message displays.
3. After reading the report, **Close** the report window.

Setting Up the Display



Before we begin placing components, select the layers and objects to view.

1. Choose the **View > Display Controls** menu item. Or, use the Display Controls stroke to display the dialog box.
2. In the Objects section of the dialog box, select to view only the following objects:

   Errors

   Geom Attributes

   Guides

   Pads

   Text

3. In the Layers section of the dialog box, select the **View Layers...** button. Ensure the following layers are visible:

   Board_outline

   Place, Placement_region_1, Place_1, Place_2

   Signal, Signal_1, Signal_2, Signal_3, Signal_4

4. **OK** the dialog box.

5. In the View Style section of the dialog box, select to view the Polygon/Trace objects as **Centerline** and Text objects as **Outline**.

6. **OK** the Display Controls dialog box.

Placing By Name

1. Choose the **[Top Menu] Placement > Select > Select By Name...** menu item. In the dialog box, enter the following and **OK** the dialog box.

   Select by: **Property**

   Select: **Components**

   From List

   Properties: **ref**

   Value: **U1**

2. Use Shift-Middle Mouse Button to zoom out slightly so you can see both the board and the U1 component placed outside the board.

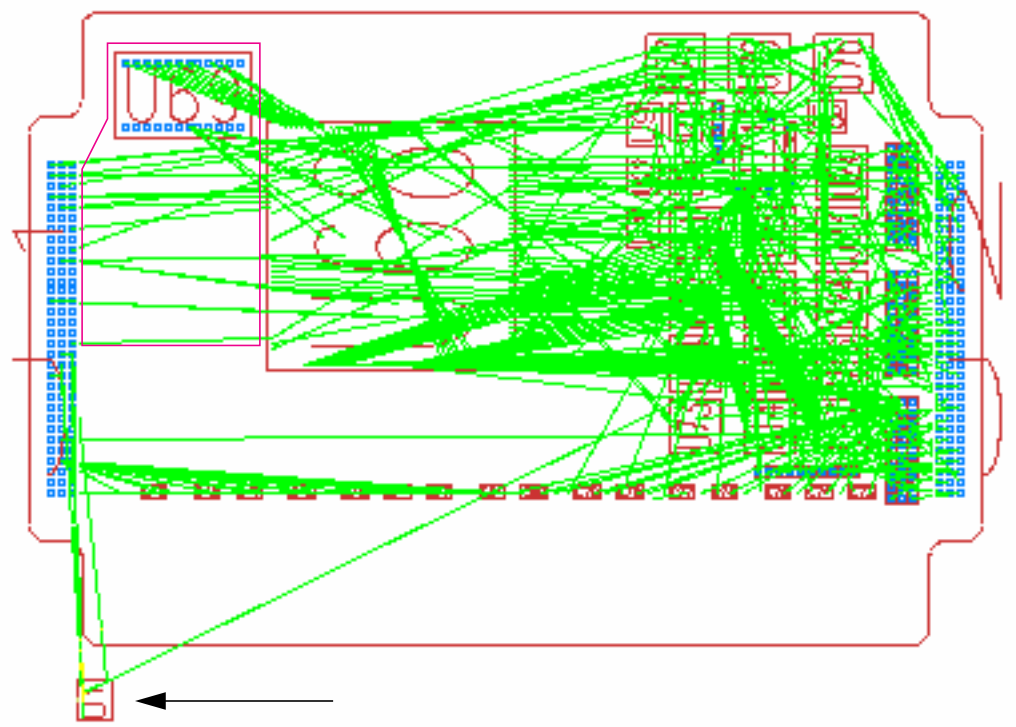


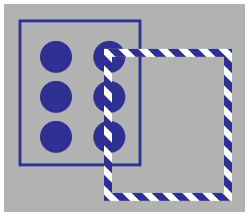
Figure 3-14. Component Ready for Placement

You also see several *guides*, which are lines showing connection between pins of the U1 component and other components on the board. The status window indicates an Invalid Placement. This is part of the automatic checking that notifies you of invalid placement and routing. Next to the Invalid Placement warning is an indicator that says Check On, meaning the automatic checking is on. In the next step, you move the component to a valid location.

3. Choose the **[Placement] Move** menu item.

   The move prompt bar displays. When you move the cursor in the edit window, a ghost or drag image of the component replaces the cursor and moves with the mouse.

   Because you selected the Connection Dynamics option in the previous lab, you see the rubber banding effect of the guides moving with the component.



4. Move the ghost image of component U1 to absolute coordinates **0.4, 2.8**. You can find the readout of absolute coordinates in the status line at the **ABS:** label.

   Click the Select mouse button to place the component. The Select Area prompt bar repeats and displays.

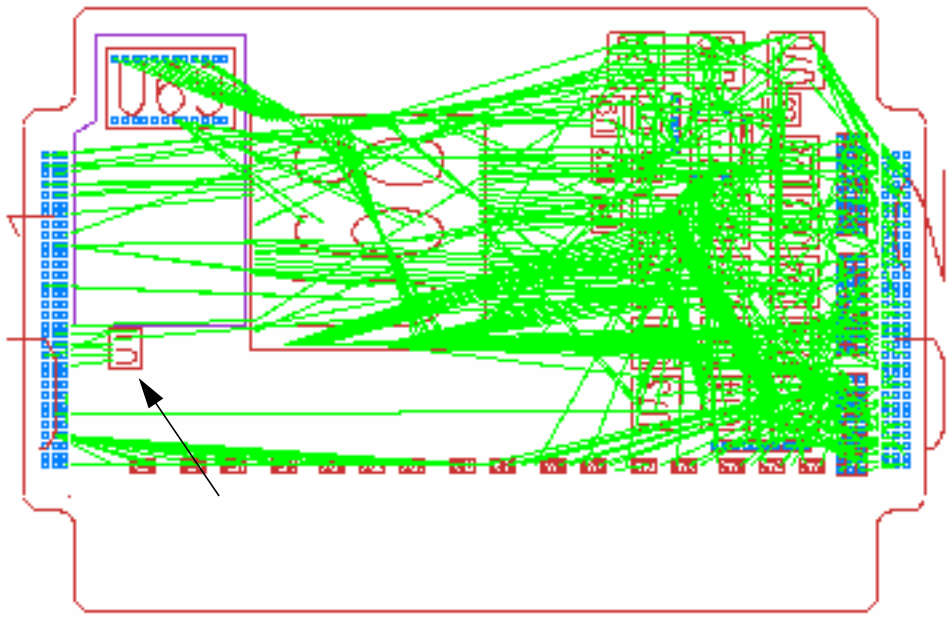


Figure 3-15. Component Placed

5. Leave the Select Area prompt bar alone, and choose the **[Placement] Select > Select By Name...** menu item again. In the dialog box, specify to select a component with a **Ref** property value of **U2**, then **OK** the dialog box.

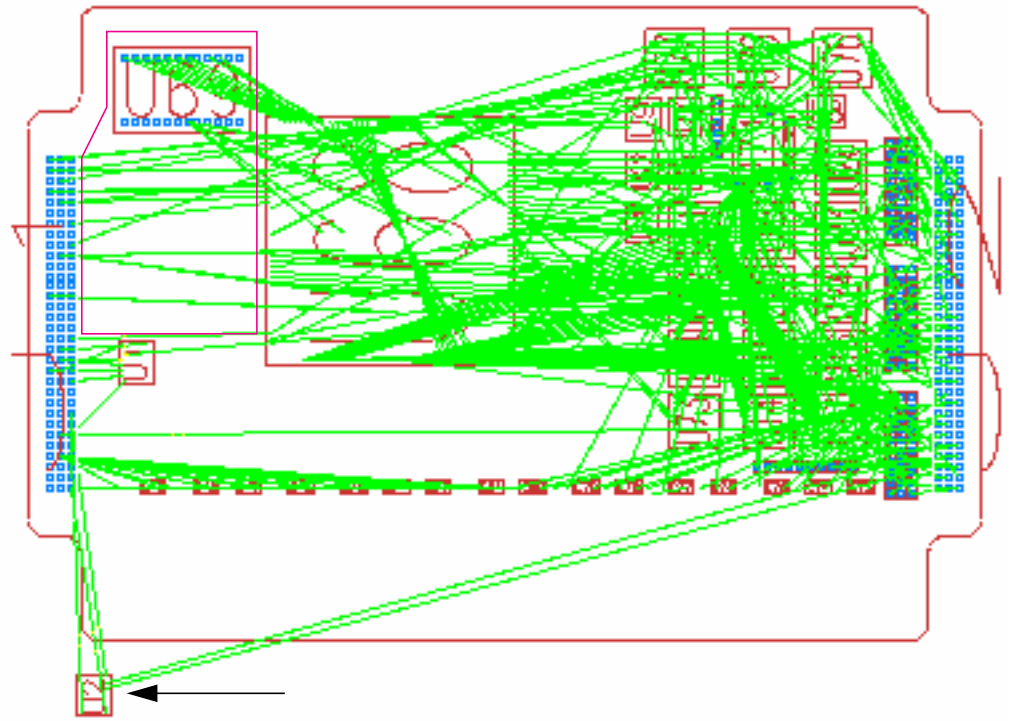
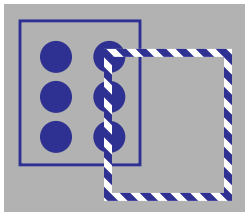


Figure 3-16. Next Component Automatically Selected for Placement

The U2 component is placed outside the board, and the connecting guides are visible. The Select Area prompt bar is still visible. It must be the only prompt bar visible.



6. Select the U2 component, and when the Move prompt bar displays, move the cursor to move the U2 component to absolute coordinates **0.8, 2.3**. Click the Select mouse button to place the component.

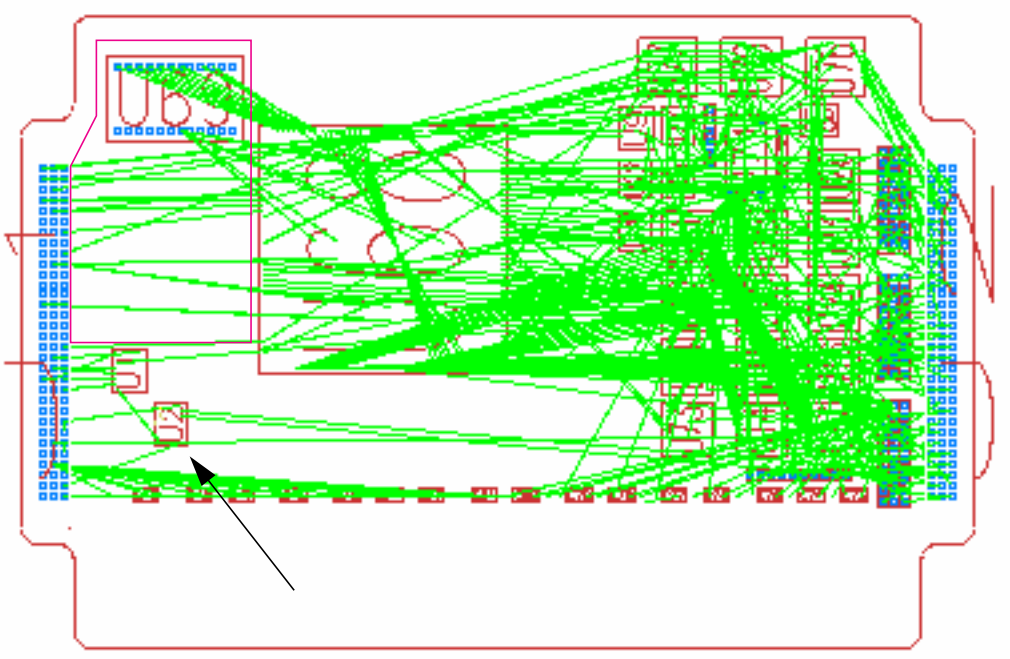


Figure 3-17. Next Component Placed

The Select Area prompt bar repeats again.

7. **Cancel** the Select Area prompt bar.

Placing from the Schematic

You can interactively place components by selecting gates shown on a schematic and dragging the cursor to the board layout. In this section of the lab, you select five gates on the schematic and place the corresponding five components on the board.

1. Display a sheet of the schematic by choosing the **View > Schematic Sheet** menu item.

2. Fill out the prompt bar as follows and select **OK**.

   Hierarchy: **/data_io/hi_speed_channel**
   Sheet: **sheet1**
   Window Setup: **left_right**

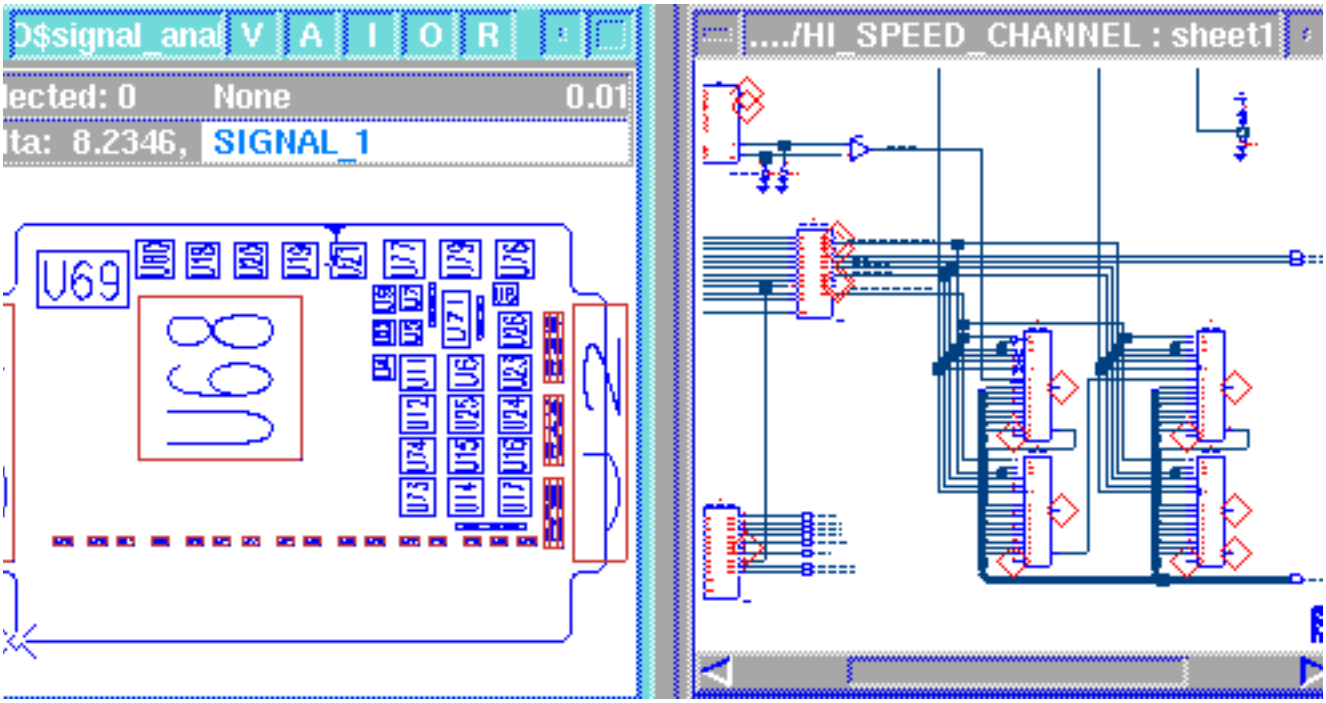


Figure 3-18. Viewing the Schematic in LAYOUT

3. Activate the schematic window by clicking on the window.

When you view the schematic from LAYOUT, remember that you are viewing the sheets through the design viewpoint. The schematic is not editable from the LAYOUT tool.

4. To select components for placement, choose the **Place Component** menu item from the Schematic window popup menu.

   A prompt bar appears and the cursor changes to a crosshair for selection.

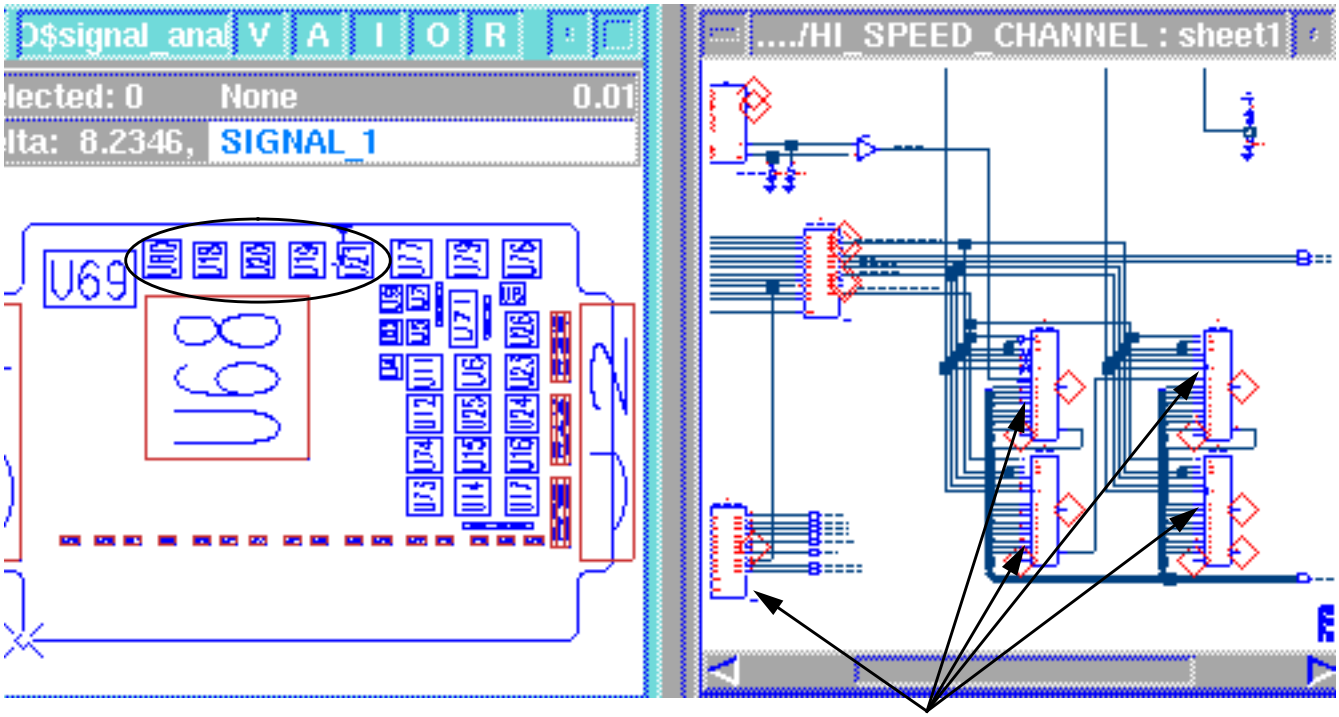


Figure 3-19. Placing Symbols from the Schematic

5. Place the cursor on one of the gates indicated in Figure 3-19 and click the Select mouse button.

6. To place the component, move the cursor to the LAYOUT Edit window. As the cursor enters the Edit window an outline of the selected component follows the cursor. Position the component at the desired location and click the Select mouse button.

   The component is placed on the board.

7. Use this same technique to place all five of the components indicated in Figure 3-19.

8. When you finish, **Cancel** the prompt bar.

9. Close the schematic window and maximize the Edit window.

Selecting Components for Placement

Now you are going to use a different method to select new components for placement.

1. Choose the **[Placement] Place Components:** menu item.

   By choosing the Place Components menu item, LAYOUT chooses a component from the queue and hangs the component on the cursor to be moved or dragged into place. As you place one component, LAYOUT chooses another to be placed. This process continues as long as there are unplaced components.

   LAYOUT selects a component for placement and prompts you for a location. If the component has the Placement_region property attached to it with a value of *analog*, place it in the analog placement area in the upper-left corner of the board. The Placement_region property value of a selected component displays next to the select count in the banner window.

You do not need to use the Move menu item to position components this time, as Place Components prompts for a location.

2. Move the cursor to position the part as you want, and click the Select mouse button.

   As soon as you click the Select mouse button, another part is selected for placement.

3. Place the components so that there is plenty of space between them for routing traces, or in case you want to reposition some of the components later.

   It does not matter exactly how the components are arranged. You'll be placing, unplacing, and autoplacing components several times. The idea is to get comfortable with the placement tools.

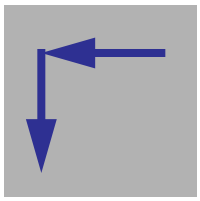
4. Place several components to see how components are automatically selected on the basis of their connectivity to components already on the board. You don't need to place the entire board, however. When you finish, **Cancel** the prompt bar.

   You must have one component selected. In the next step, you unselect it, which places it back in the queue to be placed later.

5. Choose the **[Placement] Unplace Selected Components** menu item.

   The selected component is removed, and the *Invalid Placement* warning in the status window is removed. You will place all the components later. First, you practice moving and rotating components.

Setting Up to Move and Rotate Components



1. Ensure the Select Filter is set to select only components.

   Use the Select Filter stroke (shown at left) to display the dialog box.

   When you are repositioning a component that has been placed, setting the filter ensures you are selecting only the components intended.

   Choose menu item **View >Display Controls** (or use the stroke) to display the Display Controls dialog box. Turn off the **Guides** object making it easier to see the components. **OK** the dialog box.

Moving Components

1. Examine the placement of the components you placed previously, and identify at least two components you would like to move.
2. Choose the **[Placement] Move** menu item. Place the cursor over any one of the components and click the Select mouse button.

   The outline highlights and a ghost image follows the cursor.
3. Position the ghost image of the selected component at a new location, and then click the Select mouse button to place the component. After you place the component, press the Unselect All function key (F2) to unselect all items.

   If you try to place a component too close to another component or in a keepout area while automatic checking is active, a message appears at the bottom of the window and the component remains at its original location.

   To determine if automatic checking is on, look for *Check On* or *Check Off* in the status window. To turn checking on, choose the menu item **Setup > Automatic Checking On.** To turn checking off, choose **Setup > Automatic Checking Off**. You want automatic checking off to deliberately place a component outside a placement area, or in a keepout area.
4. Repeat the **[Placement] Move** menu item (hold the Shift key down and press the Menu mouse button), and move additional components. When you are done, unselect all items and cancel all prompt bars.

Rotating Components

To make a placed section tighter and to shorten the trace lengths, try rotating components.

1. Choose the **[Placement] Rotate:** menu item.

   A select area prompt bar displays.

2. In the prompt bar, choose **Options**. In the Options dialog box, ensure only **Components** can be selected. **OK** the dialog box.

3. Place the cursor on a component you would like to rotate, and click the Select mouse button. Move the cursor, and you see the ghost image rotate about the origin of the object. Rotate the component to a new orientation, and click the Select mouse button again.

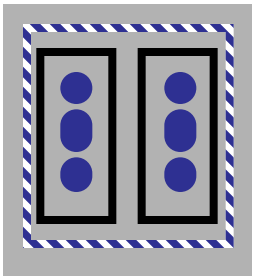
You may need to Move components to Rotate them.

If the new location conflicts with another component, is outside a placement area, or in a keepout area, the placement is not allowed with checking on. You might have to move the component to a clear area before rotating it, and then move it back into position. You can also turn checking off, which allows the violation. Rotate the component and move it to avoid the violation. Then turn checking back on.

The Select Area prompt bar repeats so you can rotate another component.

4. Experiment with rotating other components. When you are finished, unselect all components and cancel all prompt bars.

Pivoting Components



You can also pivot a component. Pivoting allows you to rotate one or more components based on their origins or centerpoints. Try using the Pivot function.

1. Choose the **Setup > Automatic Checking Off** menu item.
2. Choose the **[Placement] Pivot Components** menu item.
3. Select two components that are side-by-side.

   You can select two objects together by placing the cursor near one, holding Select area key down, and moving the cursor until the select area rectangle encloses at least part of both components and nothing else. Then release the Select mouse button.
4. Fill in the prompt bar with the following and **OK** the prompt bar.

   Angle: **90 center relative norepeat**
5. The two components pivot 90 degrees about a single basepoint.
6. Undo the pivot by choosing the menu item **[Placement] PCB Undo/Redo > Undo**.

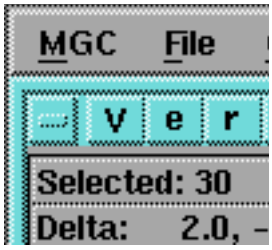
   The pivoted items are returned to their original orientation.
7. Unselect all objects and choose the **Setup > Automatic Checking On** menu item to turn on the checking again.
8. Experiment with pivoting other components. When you are done, unselect all items, correct all placement violations, cancel all prompt bars, and make sure the automatic checking is on.

Mapping Components

It is often helpful to see the component outlines of all the components before deciding which to place. In this procedure, you select the analog components and display them in an area outside the board. Then you place them.

1. Choose the **[Top Menu] Auto Placement > Select > Select By Name...** menu item. In the dialog box, enter the following information and **OK** the dialog box.

   Select by: **Property**
   Select: **Components**
   From List
   Properties: **placement_region**
   Value: **analog**



   At the top-left corner of the window note a select count of 30.

2. Choose the **[Auto Placement] Map Selected Components...** menu item. Fill out the dialog box as follows and **OK** the dialog box.

   Top Only
   Map Components to Area: **Specify Area**

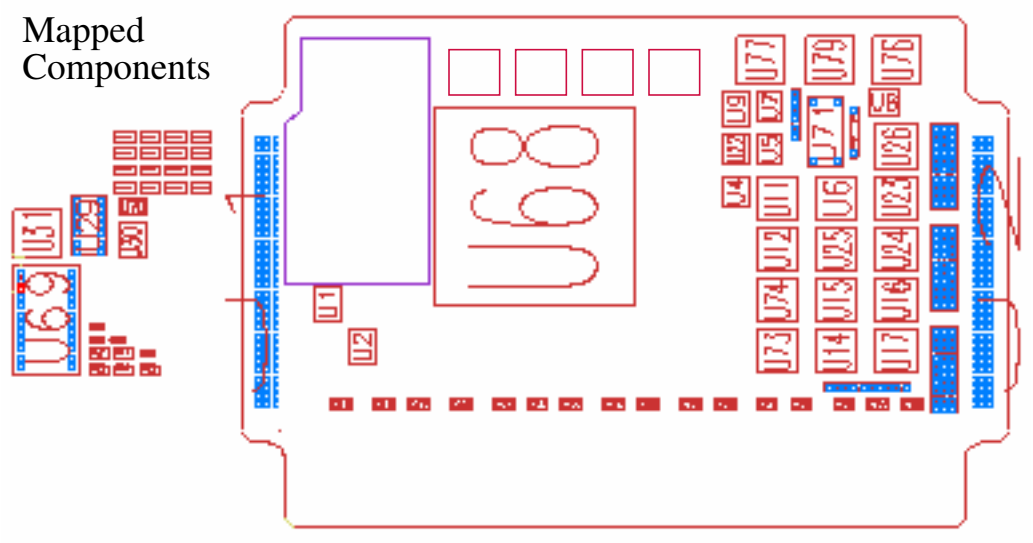


Figure 3-20. Mapped Components

Map components

Drag components onto board

3. Using the Select mouse button, define a rectangle outside the board outline.

   The analog components display in the area you defined. Even though Map Components is an Auto Place function, this feature is very useful in interactive placement.

4. Use the Move, Rotate, and Pivot commands to place the analog components on the board. Place the components in the analog placement region in the upper-left section of the board.

5. While placing components on the board, try these shortcuts to rotate and flip components. In the middle of the Move function:

   - Shift—Right Mouse Button rotates the component.
   - Ctrl—Right Mouse Button flips the component.

You do not have to be very exact about the location of each component, because you are going to use another method to place them again later.

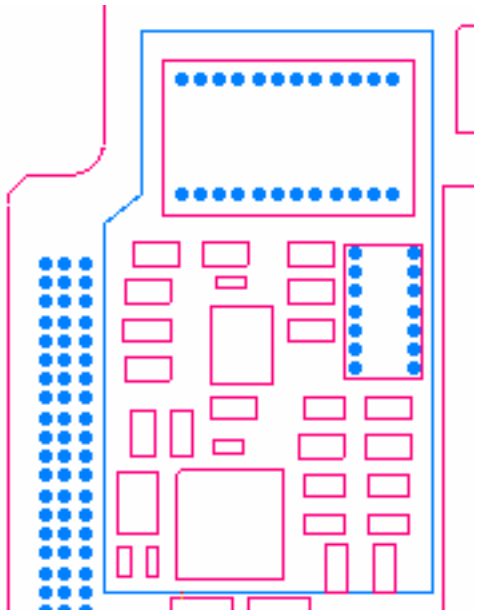


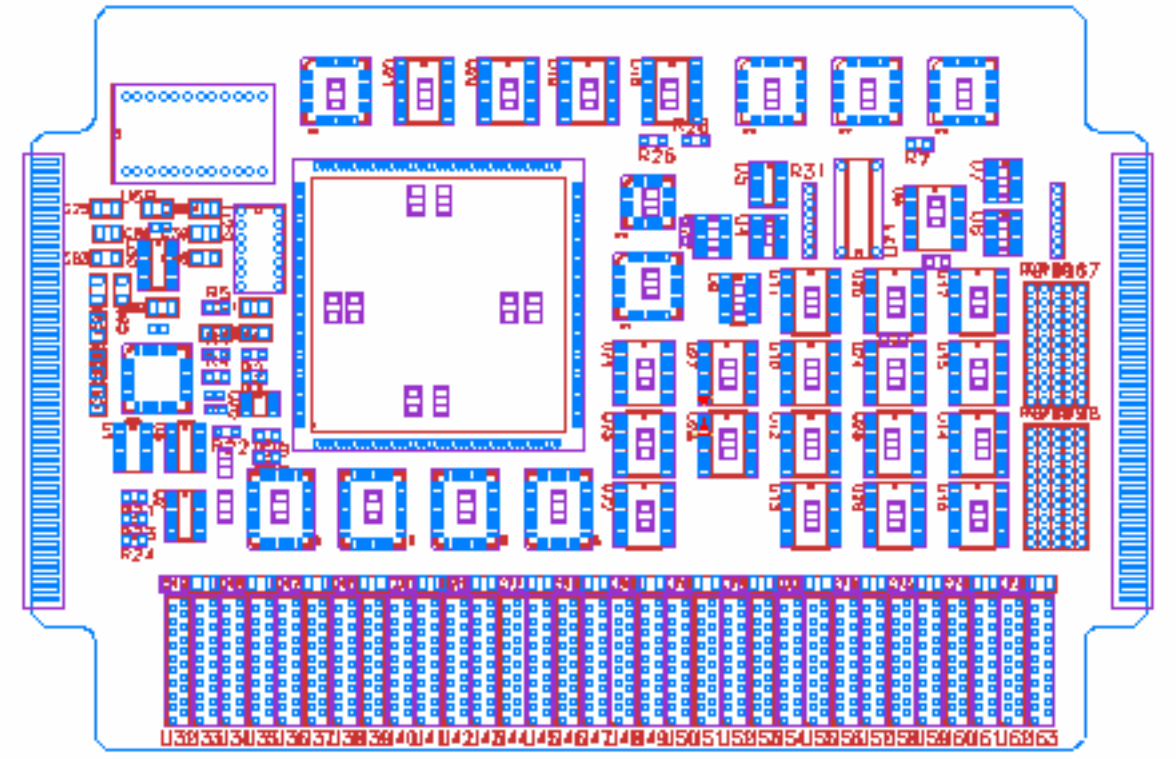
Figure 3-21. Analog Section of the Board

The autoplacer automatically places components with a placement region property in the placement region having the matching value. Interactive placement does not enforce placement regions.

Congratulations! You have completed the "Interactive Placement" lab exercise. Continue with Lesson 4: "Automatic Placement".

Lesson 4
Automatic Component Placement

This next lesson in the Placement and Routing modules provides an introduction to the major features of the LAYOUT tool. The Placing Components on a Circuit Board module acquaints you with the interactive and automatic placement features, and prepares you for the next module, Routing. The Automatic Component Placement lesson discusses placing components on the board automatically.



Objectives

In this last lesson of the component placement module you examine:

- Setup of the automatic placement tools.
- Control of the automatic placement tools.

In this lesson, you also examine other aspects of component placement including:

- Setup considerations for automatic placement.
- Setup parameters for automatic placement.
- Techniques for automatic placement.
- Techniques for optimizing component placement.
- Relationship between histogram data and routing density.
- Relationship of the report routability data to potential router performance.

Set Up Controls for Automatic Placement

Placement sites and boundaries allow you to place controls on automatic placement.

- **Define Placement Sites**

  You can generate placement sites (selected locations) for constructive placement of components. Larger components can be placed that way, then the sites can be cleared for placement of all other components.

- **Define Placement Boundary**

  You can define a temporary boundary within the placement area for constructive placement. The boundary can be used with or without placement sites.

Defining Placement Sites

You can specify a grid of placement site points or a matrix grid, for placement purposes only. Placement sites facilitate the automatic placement of large components. You can specify different values for the x and y axes to create any site grid.

To ensure that all of these placement sites coincide with the placement grid, the grid you specify should be a multiple of the current placement grid.

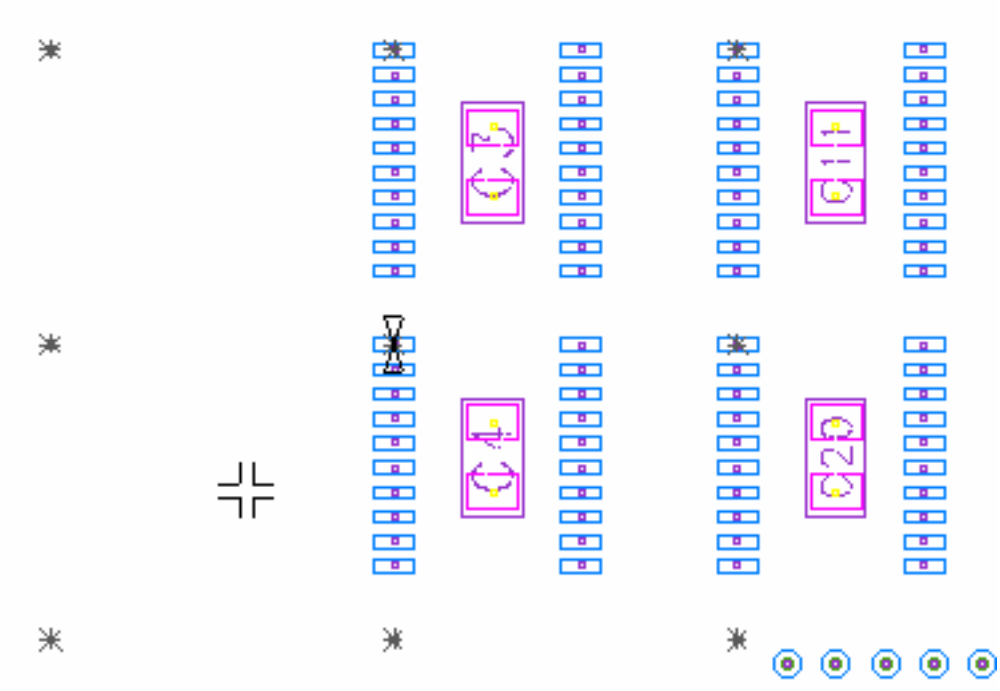


Figure 4-1. Components Placed on Placement Sites

Defining Placement Boundaries

The board geometry requires a placement outline. The placement outline defines the area on a board into which components can be placed. You can define another rectangular area, called a placement boundary, to further confine automatic placement. There can be only one placement boundary defined at a time for your board. Defining a placement boundary automatically replaces any existing boundary.

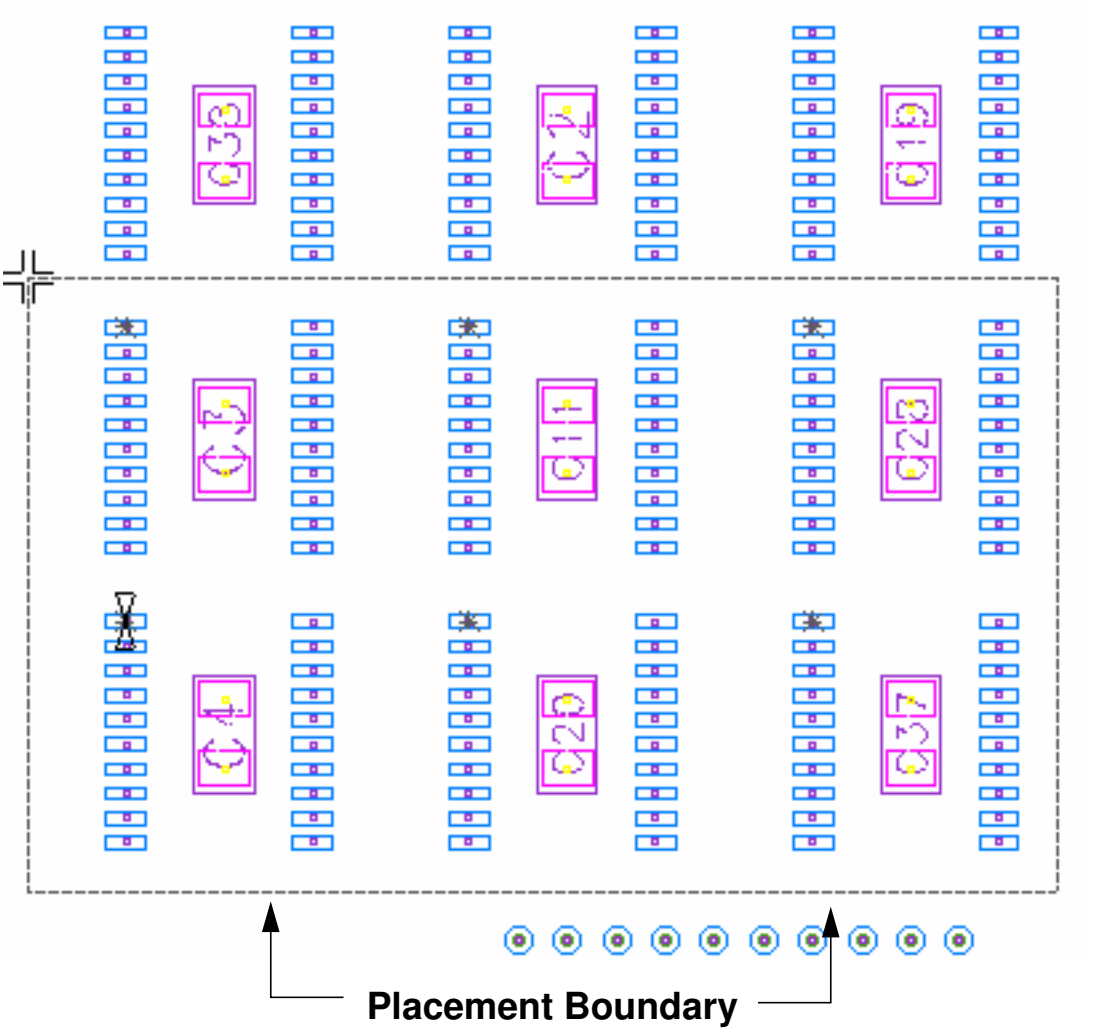


Figure 4-2. A Placement Boundary Confines the Placement Area

Automatic Placement

You can use automatic placement features to place just a few components or all components at once.

You can do any of the following:

- Auto Place Selected
- Auto Place All
- Auto Place Some

The autoplacer selects components whether they are unplaced or mapped off the board. If you choose to place only pre-selected components, these are placed automatically by connectivity as well.

Auto Place Selected

Choosing to autoplace selected components provides the following options:

- Automatically place a group of components.
- Allow the autoplacer to place components in any valid orientation to minimize the placement area and achieve the shortest possible net lengths.

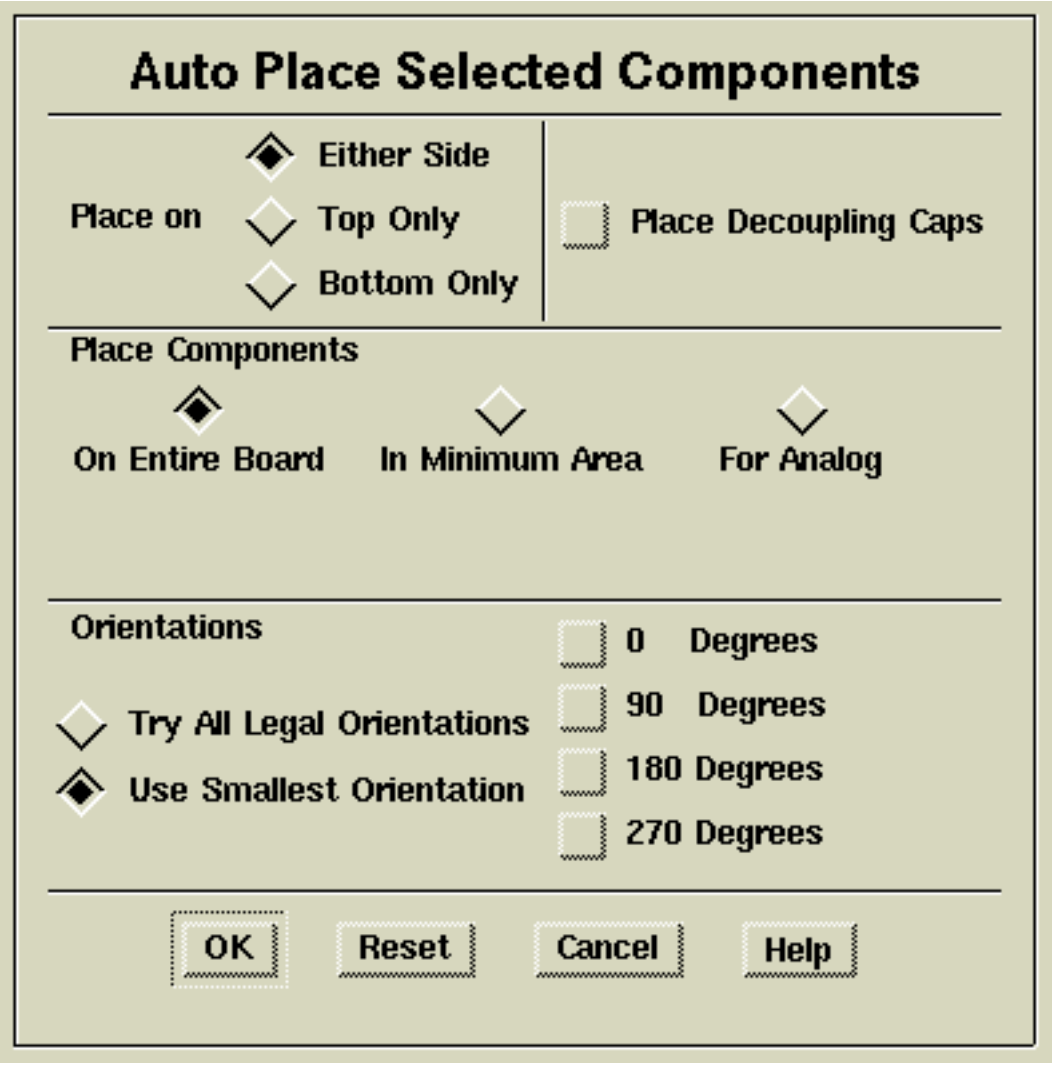


Figure 4-3. Auto Place Selected Components Dialog Box

You can use the select by name feature to select all the components in a predefined group, or to select all components with a specific property and value. Once you have selected the components, you can place them automatically as follows:

- **On Entire Board**—uses the entire placement area.
- **In Minimum Area**—autoplacer places the selected components in the smallest rectangle that can contain them. The autoplacer rotates the components to achieve the smallest area.
- **For Analog**—autoplacer places the selected components in the smallest rectangle that will contain them and orients them for minimum net length. The autoplacer does even more rotating and minute tweaking to place components in the smallest area.

Auto Place Some or All

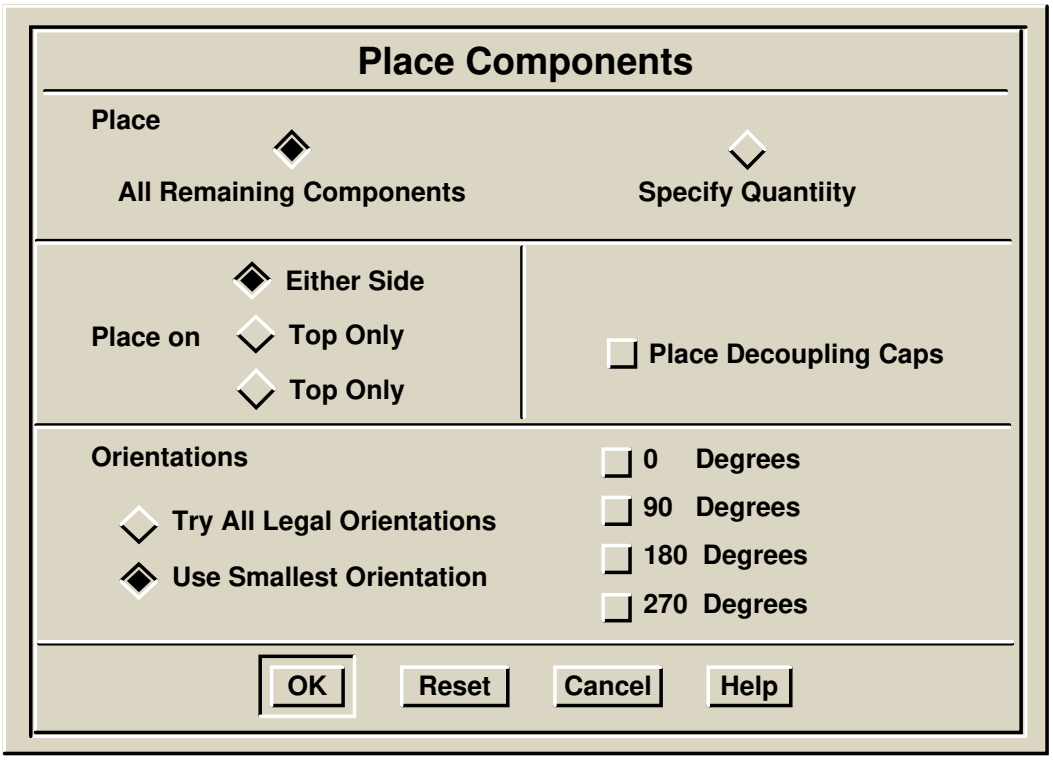


Figure 4-4. Place Components Dialog Box

- **All Remaining Components**—places all unplaced components and any components mapped off the board.
- **Specify Quantity**—enter the number of components to place. The auto placer places that number in descending order of connectivity.

  Specifying the quantity of components to place is the most effective method. Once the components are placed, use interactive placement functions to make any adjustments. Then use Place Components to place another set of components.
- **Either Side/Top Only/Bottom Only**—indicates the side of the board on which to place components.
- **Place Decoupling Caps**—places unplaced or mapped components and their attached decoupling capacitors in an alternating sequence.
- **Try All Legal Orientations**—attempts to place each component in any valid orientation.
- **0, 90, 180, 270 Degrees**—specifies any or all possible placement orientations.

Automatic Placement Improvement

Automatic swapping improves a placement by exchanging the positions of pairs of swappable components, gates, pins, and pinsets when the exchange would result in a reduction of net length.

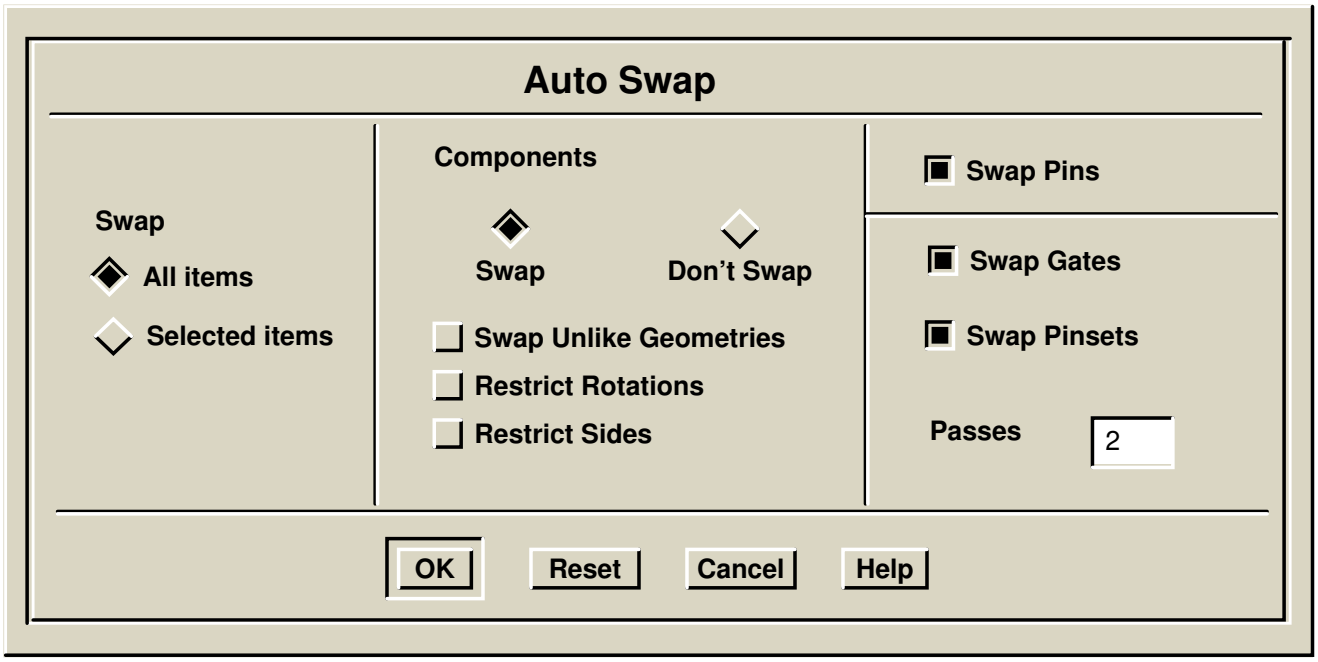


Figure 4-5. Auto Swap Dialog Box

You can use the Auto Swap dialog box to set up and control automatic swapping of all swappable items on your board.

- **All Items**—considers all swappable items as candidates for swapping.
- **Selected Items**—limits swapping to selected components, gates and pins on selected components, and selected pinsets.
- **Swap/Don't Swap**—enables/disables component swapping.
- **Swap Unlike Geometries**—swaps components regardless of geometry. Otherwise, only components of the same geometries are swapped.

- **Restrict Rotations**—only swaps components placed at the same orientation.
- **Restrict Sides**—only swaps components on the same side of the board.
- **Swap Pins**—swaps eligible pin pairs.
- **Swap Gates**—swaps eligible gates.
- **Swap Pinsets**—swaps eligible pinsets.
- **Passes**—sets the number of swapping passes for gates and pinsets.

Before you auto swap components, be certain to protect connectors and testpoints.

Interactive Swapping

You can trade positions of any two selected components, any two gates on the board that have matching swap codes, or any two pins within the same gate that have matching swap codes.

To perform the swapping:

1. Set the Select Filter to the object type: components, gates, or pins.
2. Select the two component, gates, or pins you want to swap.
3. Choose the appropriate swap menu item from the edit window popup menu:

   [Placement] Swap > Swap Components

   [Placement] Swap > Swap Gates

   [Placement] Swap > Swap Pins

   [Placement] Swap > Swap Pinsets

Changing Part Numbers, Padstacks, and Geometries

During placement or routing in LAYOUT, you might need to change a part number or modify a geometry for a component.

You can change part numbers, component geometries, and padstack geometries without leaving LAYOUT. However, you can only make part number or geometry changes that do not affect the logic of the design. Also, you cannot change the board geometry within LAYOUT.

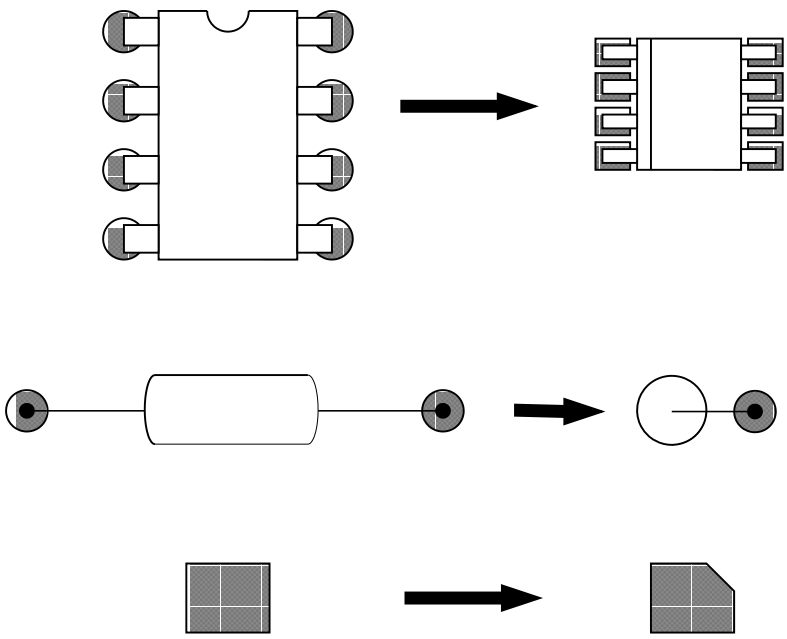


Figure 4-6. Examples of Alternative Part Numbers and Geometries

You must plan ahead for changing part numbers, padstacks, and geometries.

- Alternate geometries must be in the *geoms* design object.
- Alternate part numbers must be in catalogs referenced in the *catalogs* design object.

In the process of changing a part number, the part geometry changes along with the new part number. For example, you would change part numbers to switch a component from a DIP package with one part number to a surface mount package with another part number. The new part number must appear in a catalog listed in the *catalogs* design object for the design.

Changing Component Geometries

Changing component geometries is more restrictive than changing part numbers because the component's part number doesn't change. For example, a resistor might have two versions of the same part number: one with a geometry for placing the resistor upright and one for placing the resistor on its side. To switch between these two geometries, you would change geometries rather than change part numbers.

Changing Padstack Geometries

Changing padstack geometries is independent of part numbers. You can change the padstack for a pin to any available padstack and the new padstack overrides the original padstack without affecting the component geometry. The new padstack geometry is recorded in the *comps* design object. A typical use of this feature is to substitute a smaller padstack at a pin so that you can squeeze a trace past the pin.

Changing Part Numbers and Geometries

When changing part numbers or geometries in LAYOUT, the *geoms* design object for the design must contain the new geometries. If the new geometries do not exist in the *geoms* design object, you must add the geometries and update the *geoms* design object in LIBRARIAN. However, you do not have to end the current LAYOUT session; you can read the updated *geoms* design object from within the current LAYOUT session.

Moving Pins

You can select a pin of a component and move the pin to a new location. The new location is stored relative to the component so that if you move the component, the pin moves with it, maintaining its offset. Components whose pins you want to move must have the Component_pins_moveable attribute assigned to their geometries. The Component_pins_moveable attribute is assigned when you create the geometry in LIBRARIAN.

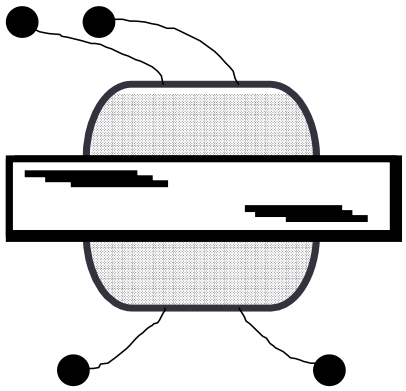


Figure 4-7. Component Pins are Moveable

You should only move pins in very specific instances. For example, you should never move the pin of a pre-packaged through-pin component. The most common use of this feature is for moving the pins of components with flexible leads, as in the case of transformers. Changing the geometry is recommended for situations where a component does not have flexible leads.

To move pins on a component, choose the menu item:

Routing > Extended Menu > Move Pin

Interactive Placement Cleanup

Following automatic placement you might want to adjust the position and orientation of components. LAYOUT provides interactive features to help you do this.

- Align Components in Rows and Columns
- Pivot
- Swapping

Automatic checking ensures that any changes you make to the placement are within the design rules for your board. If you attempt to make a change that would violate a design rule, the change is not allowed.

Aligning Components

LAYOUT provides a feature that precisely aligns a row or column of components, with the alignment based on pins, component origins, or component edges. Components are moved only vertically for row alignment and only horizontally for column alignment. Preselection is not required. If no components are selected, you are prompted to select components within an area before proceeding. You must select at least two components. You can select the components after you choose the function.

Choose the **Placement > Extended Menu > Align Components...** menu item from the Edit window popup menu.

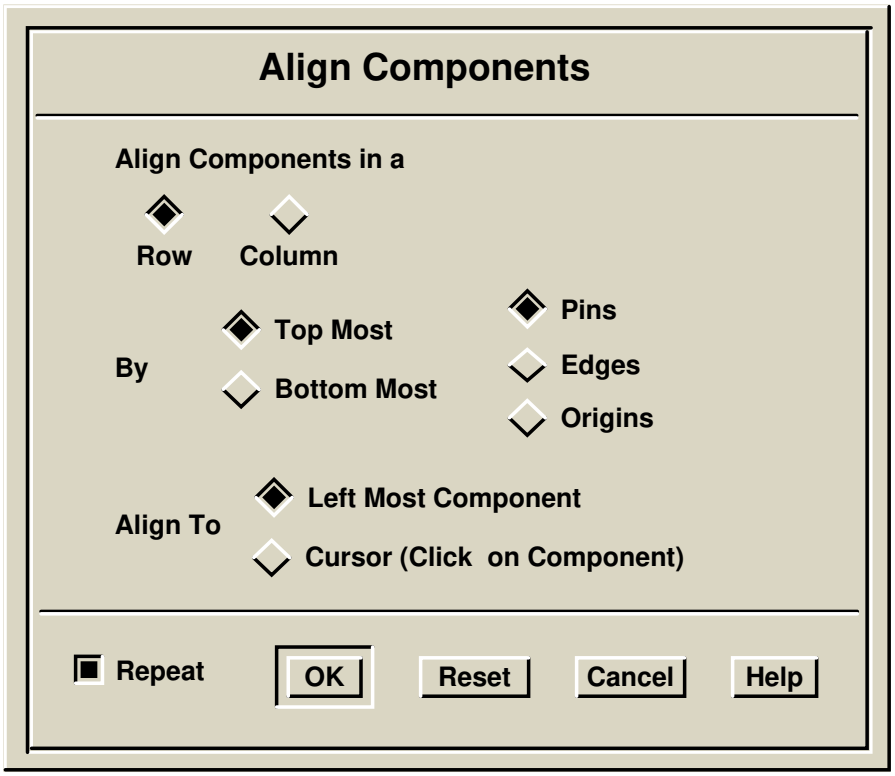


Figure 4-8. Align Components Dialog Box

If LAYOUT prompts you to select components, but no components highlight when you define the selection area, press the Options button on the Select Area prompt bar. In the Select Area (Options) dialog box that displays, press the Components button to temporarily set the selection filter to components.

Adding Discrete Components

Sometimes layout considerations require the addition of more decoupling capacitors than the schematic originally provides. You can add decoupling capacitors to a design in LAYOUT. The following rules summarize the requirements for adding decoupling capacitors:

- Only part numbers in catalogs referenced in the *catalogs* design object are available.
- Available part numbers are limited to two-pin discrete components with geometries that are included in the *geoms* design object.
- Added decoupling capacitors can only be connected to global nets. Global nets are the power and ground nets on the board.
- A global symbol with a name that exactly matches the global net name must exist in the symbol search path for successful back annotation.
- Both the symbol name and the symbols's Comp property must match the Comp property of the discrete component on the board for successful back annotation.

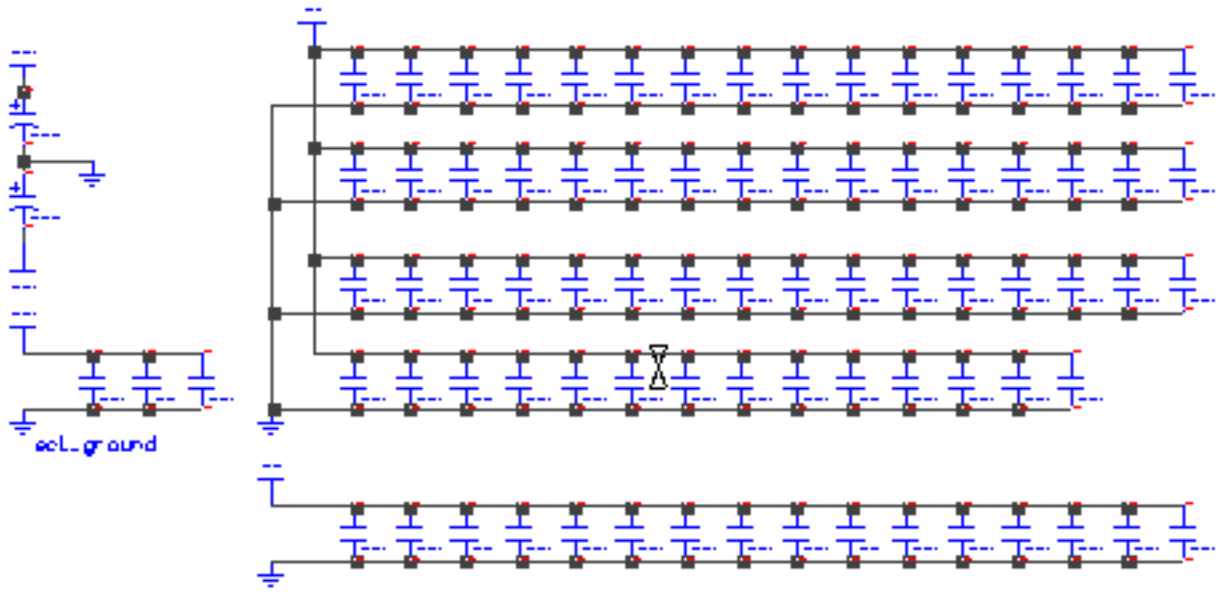
Adding discrete components to the design is a two-part process. First, you select one or more of the available part numbers. Second, you assign a global net to each physical pin and specify the number of components to add. Optionally, you change the reference designator prefix for the added components. After completing the process of adding discrete components to the design, you place the added components on the board using interactive or automatic placement procedures.

Back Annotating Discrete Components

After adding discrete components to your board, you can control how the components are back annotated to the schematic. The **Setup > Schematic > Discrete Annotation** menu selection allows you to set up parameters for annotating the discrete components. You can define the following:

- Schematic sheet onto which the discretes annotate.
- Units of the sheet.
- Symbol search path where LAYOUT looks for discrete symbols.
- Distance between gates on the sheet.

Selecting menu item **File > Back Annotate** update the viewpoint and creates the specified sheet with the discrete gates that you added, just as if the sheet was created in Design Architect.



This menu selection also back annotates other changes to the physical design, such as an altered component location, a changed reference designator, or an added user-defined property.

Thermal Analysis

Computer-assisted thermal analysis simulates heat transfer within a PCB design and its thermal interaction with the surrounding environment.

Thermal analysis models:

- Heat dissipated by components.
- Heat transfer through components to the board.
- Heat transfer to and from the environment through conduction, convection, and radiation.

AutoTherm evaluates the thermal environment and heat transfer process to determine whether electronic parts will operate within an acceptable temperature range. Thermal analysis produces design information about how heat is generated and dissipated. A designer can use this information to help ensure that the reliability of the electronics will not be degraded.

Prerequisites for Thermal Analysis

For a design to be evaluated by AutoTherm, thermal properties must be present to give the analysis routines needed information. The additional AutoTherm properties must exist on the schematic symbols and in the part number descriptions in the catalog file. Mentor Graphics supplied libraries already have thermal properties included.

Required properties for evaluating conduction heat transfer are:

- Therm_r
- Pow_typ

Required properties for evaluating convection heat transfer are:

- Therm_r
- Therm_jc
- Pow_typ
- Comp_height
- Surface_area

For a complete listing of the thermal properties, see the AutoTherm User's Manual.

Thermal Analysis Process Flow

After transferring the design to AutoTherm, you perform a thermal evaluation. You then decide whether to change the placement and re-evaluate or to transfer the design back to Board Station.

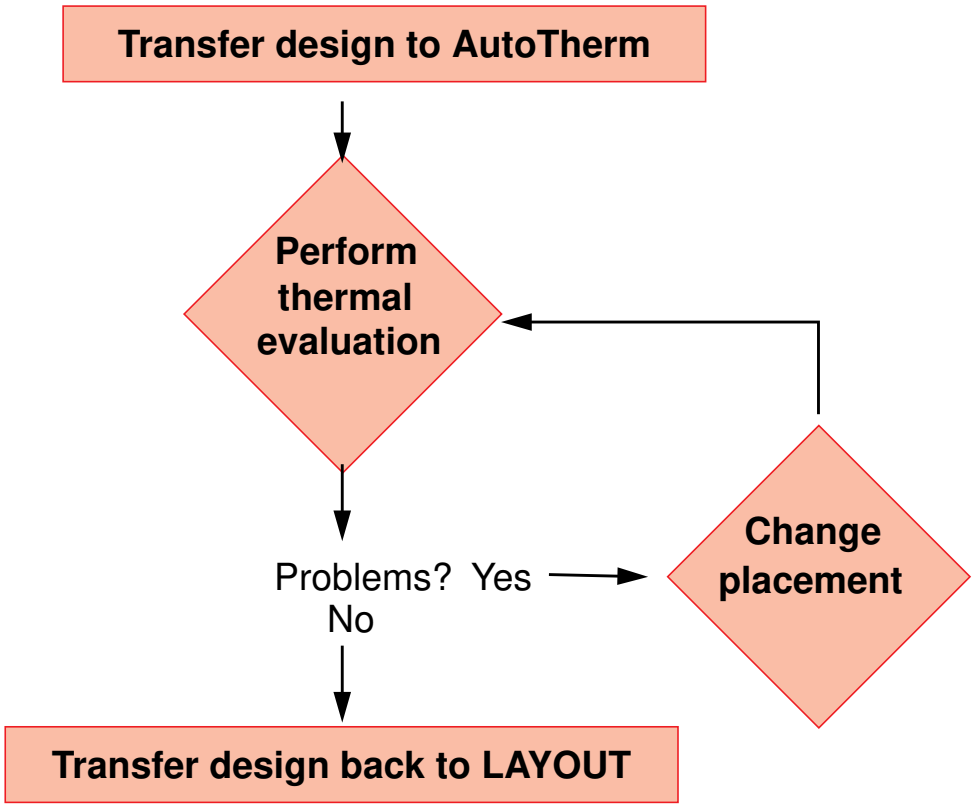


Figure 4-9. Thermal Analysis Process Flow

Placement Evaluation Tools

LAYOUT provides two features that can help you understand how your placement might be improved to make it easier to route:

- A histogram of estimated routing density.
- A routability predictor.

You can use these tools with a partial or full placement.

Placement Histogram

The routing density histogram projects trace density on the board based on the default padstack size defined in the board geometry.

To view the histogram, choose the **Report > Placement Histogram** menu item. Two histogram displays appear in the Edit window; one to the right of the board outline, and a second above it. The top histogram shows density of horizontal routes; the one to the right shows density of vertical routes.

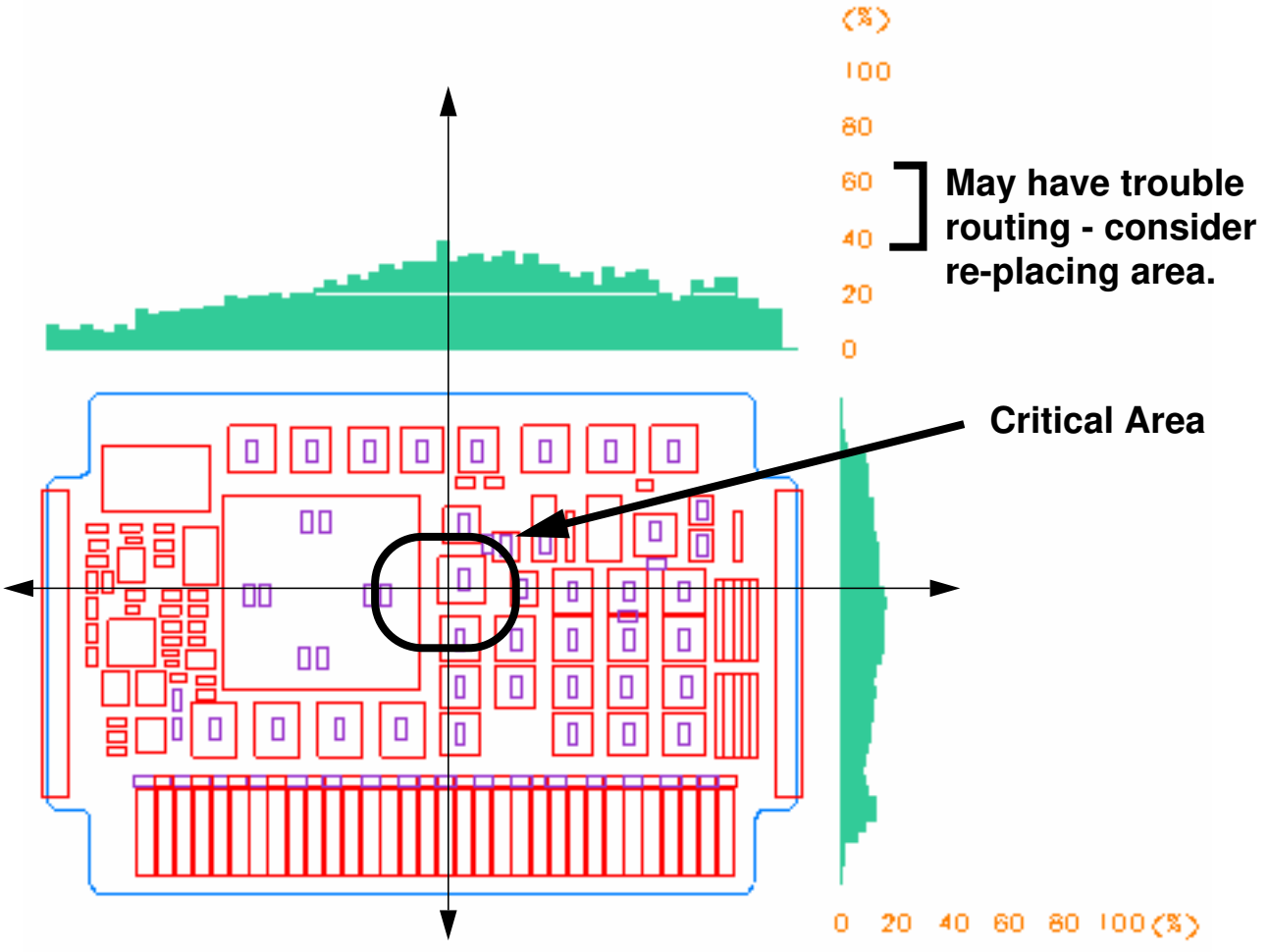


Figure 4-10. Histogram Report

The peak indicates a 40% utilization of available routing channels on the X-axis.

Routability Predictor

The routability predictor is a routing simulator that calculates routing capacity through specific areas of the board it defines as global cells. The results of the calculations display as a density map. The global cells are individually assigned one of five density layers, depending on the percentage of routing capacity used. The density map shows areas of potential routing congestion. You can make placement adjustments to achieve a more even potential distribution of traces over the board.

You invoke the predictor by choosing the **Report > Routability** menu item.

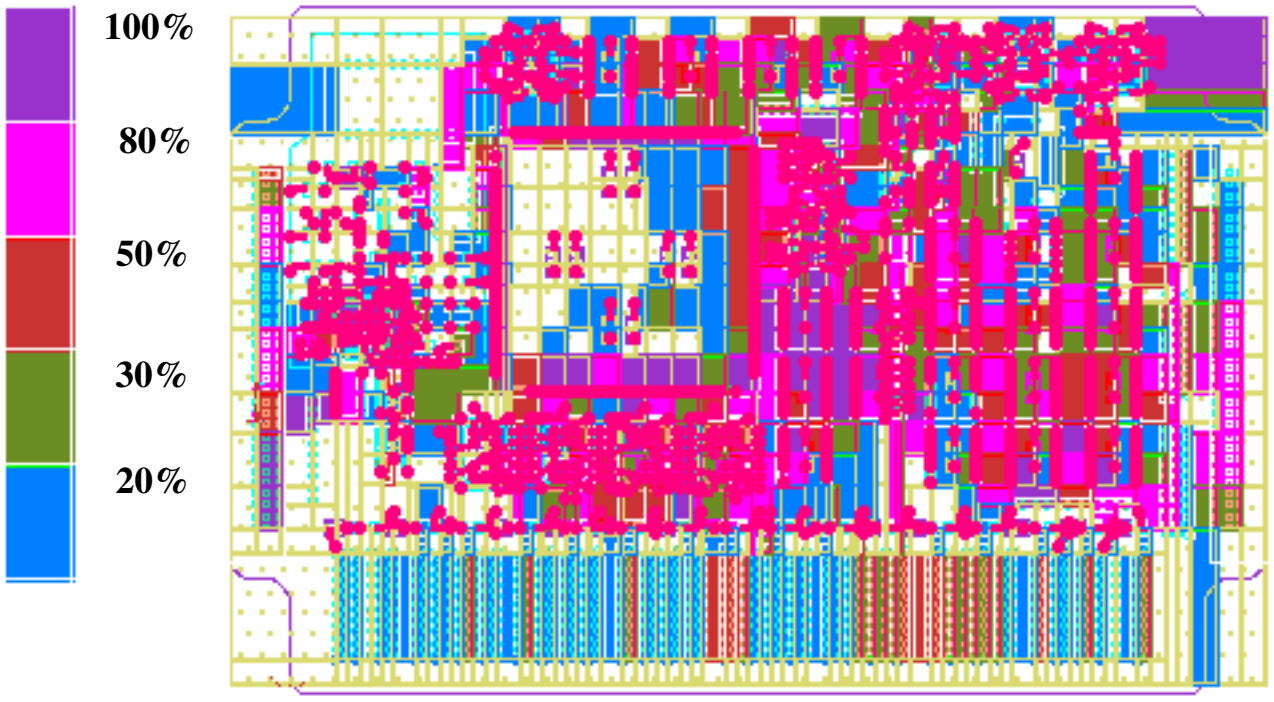


Figure 4-11. Density Map

This map was calculated on density layers 1 through 5, using density values of 100%, 80%, 50%, 30%, and 20%, respectively. This means the global cell that displays on layer Density 4 represents the percentage of used routing capacity that is greater than 30%, but less than or equal to 50%.

Summary

This module provided an introduction to the major features of component placement. In this module, you gained an understanding of how to control placement with attributes, properties, and setup procedures. You also learned both interactive and automatic placement techniques.

In the next module, you will learn routing concepts and practice both interactive and automatic routing procedures.

Lab Exercise

In the lab exercise you use several automatic tools to place components on the circuit board. You continue the placement process by optimizing the placement of component locations and the assignment of gates and pins. This swapping process minimizes the routing congestion and improves routability. To verify the quality of the placement you display a histogram of routing channel density and a display of board routability.

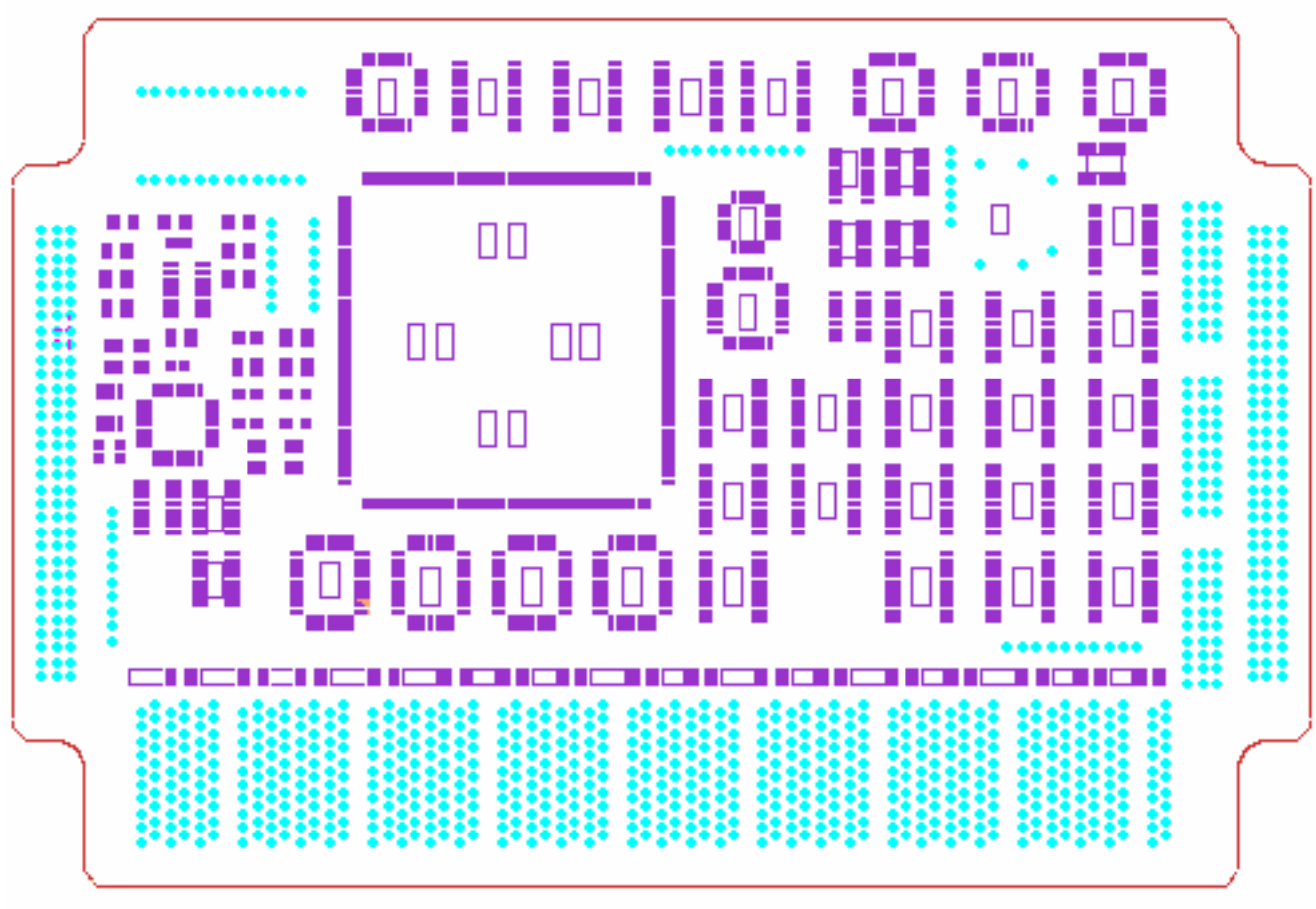
Upon completion of this lab exercise you should be able to:

- Unplace all components except fixed or protected components.
- Automatically place components into regions created in the last lab.
- Improve placement by swapping components, gates, and pins.
- Examine routing channel density using the Histogram.
- Examine routability using the Report Routability function.

Turn to Module 5—Lab 4: "Automatic Component Placement".

Lab 4
Automatic Placement

This lab exercise familiarizes you with several automatic tools to place components on the circuit board. You optimize the placement of component locations and the assignment of gates and pins. This swapping process minimizes the routing congestion and improves routability. To verify the quality of the placement, you display a histogram of routing channel density and a display of board routability.



Introduction

In this lab exercise, you use several automatic tools to place components on the circuit board. Then, you continue the placement process by optimizing the placement of component locations and the assignment of gates and pins. This swapping process minimizes the routing congestion and improves routability. To verify the quality of the placement, you display a histogram of routing channel density and a display of board routability.

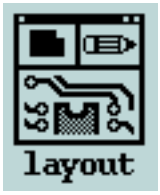
Upon completion of this lab exercise you should be able to:

- Unplace all components except fixed or protected components.
- Automatically place components into regions created in the previous lab.
- Improve placement by swapping components, gates, and pins.
- Examine routing channel density using the Histogram.
- Examine routability using the Report Routability function.

Procedure

In this lab exercise, you use the LAYOUT tool to automatically place components on the board and optimize the placement.

Preparation for Lab



1. Invoke the Design Manager. Invoke LAYOUT from the tools window. Use the navigator to open LAYOUT on the **sig_az** design.

   If an Invocation Switch dialog box appears, select the **OK** button without selecting the switch.

   A Report-Startup message might appear.

2. Close the report window.

Removing Existing Components

In the previous lab, you placed components on the board interactively. Before you can begin this exercise of automatically placing components, you need to remove all components from the board except the connectors that are fixed in their locations, the MCM, the row of capacitors, and the group of resistors, all of which are protected.

1. Choose the **[Top Menu] Auto Placement > Unplace All Components** menu item. In the Unplace All Components confirmation dialog box, select **OK**.

   This action removes any unprotected components and places them back into the placement queue. The connectors are not protected, but were fixed in their locations when the board was created in LIBRARIAN. Fixing has the same effect as protecting.

   You might find it easier to place the components with the Guide layer turned off. The guides can reduce visibility.

2. If the guides are visible, choose menu item **View > Display Controls**. In the Display Controls dialog box, turn off the **Guides** object.

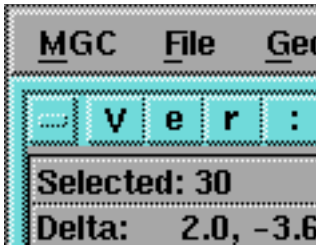
Placing a Circuit Group

Previously, you created a Placement Rule area and identified an associated circuit group named *analog*. In the next several steps, you autoplace the analog circuit group.

First, you select the components having the Placement_region property with a value of *analog* attached.

1. Choose the **[Auto Placement] Select > Select By Name...** menu item. In the dialog box, enter the following, then **OK** the dialog box.

   Select by: **Property**
   Select: **Components**
   From List
   Properties: **placement_region**
   Value: **analog**



At the top-left corner of the window you should see the select count equal to 30.

Next, you automatically place the analog components.

2. Choose the **[Auto Placement] Place Selected Components...** menu item. In the dialog box, select the following:

   Place on: **top only**
   Place Components: **In Minimum Area**
   Orientations
   Try All Legal Orientations
   Click on all four orientations to make them legal.

3. **OK** the dialog box.

   The analog components are automatically placed one-by-one.

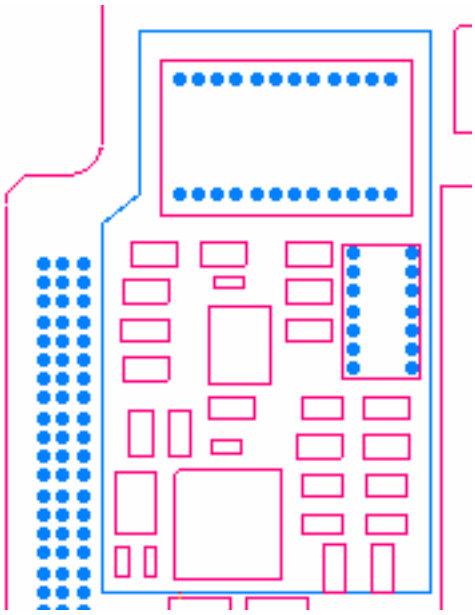


Figure 4-12. Analog Section Automatically Placed

4. After the analog components have been placed, make a note of the Total Manhattan Length displayed in the message window at the bottom of the session window.

   Next you are going to unplace the components again, and then re-place them using a different option.

5. Choose the **[Auto Placement] Unplace Selected Components** menu item.

6. Select the analog components again by name, as you did in step 1.

7. Place the selected components again as you did in step 2, except when you fill in the Place Selected Components dialog box, choose the option **For Analog**. **OK** the dialog box.

   The components are again placed one at a time in the analog placement area.

8. Leave the components selected so you can more easily protect them later.

9. Note the placement and the Manhattan length this time.

   Although the Manhattan length (a measure of how far it is between pin connections) is different, and maybe even longer than before, the space between components is greater, so you can more easily route these components.

   Next, you protect the analog section so that the components cannot be accidentally moved or unplaced.

10. Choose the **Setup Placement > Protect Components** menu item.

    The selected components are protected and unselected. The color of the components has been changed to indicate they are protected. If you want to modify the placement of these protected components, you must first unprotect them.

Testing Autoplacement

Next, you have LAYOUT automatically place the remaining components.

1. Choose the **[Auto Placement] Place Components...** menu item. In the Place Components dialog box, choose **Top Only**. Ensure the other options are set as follows:

   Place: **All Remaining Components**
   Place on: **Top Only**
   Orientations: **Use Smallest Orientation**
   Click on all four orientations.

2. **OK** the dialog box.

Use controls for effective autoplacement.

The remaining components are placed, but their placement might not be very good, as the components are scattered about with no order or organization. Some of the digital components might even be in the analog placement area.

Next, you unplace the board again. Later, you define additional placement areas for some digital parts and try placing them again.

3. **Unplace All Components** from the board.

   You can safely unplace all components without unplacing the analog components, because they are protected.

Adding Placement Controls in LAYOUT

Currently, only the analog parts have a Placement_region property, But suppose as you are laying out the board, you decide you want all the RAM chips placed together in a specific location. You can add a placement region property to the components in LAYOUT, define a placement region, and automatically place the components.

Select Unplaced Components

1. Choose menu item **[Auto Placement] Select > Select by Name**.

   The Select By Name dialog box displays.

Select by Name has many options to aid precise selection.

2. Select the following buttons in the dialog box.

   Select by: **Property**
   From List
   Select: **Components**

3. In the Properties list box, select the **Comp** property.
4. In the Value typing box, enter: **AM2168**
5. **OK** the dialog box.

You now have 32 items selected.

Add the Placement_region Property

1. Choose menu item **Properties > Component Properties > Placement Region**.

   The Add Placement Region Property dialog box appears.

2. In the dialog box, enter: **memory**.
3. **OK** the dialog box.

Create the Placement Rule Area

Placement rules areas control the autoplacer.

Now you define the placement area for the AM2168 components.

1. Choose the **[Top Menu] Placement > Add Placement Area > Add Placement Rule Area...** menu item. In the dialog box, enter the following, and then **OK** the dialog box.

   Side: **Top**
   Fill Pattern: **3**
   Maximum Height: **[Leave this box blank]**
   Minimum Height: **0.0**
   Circuit Group: **memory**

2. Using the Select mouse button, define the four points of the memory placement area shown in Figure . The coordinates given are absolute coordinates. **OK** the prompt bar.

Alternatively, you can enter absolute coordinates by typing in the Edit window. Type **coo** *for coordinate. Then type the coordinate location and hit a carriage return. For example:* **coo 1.7 5.75**

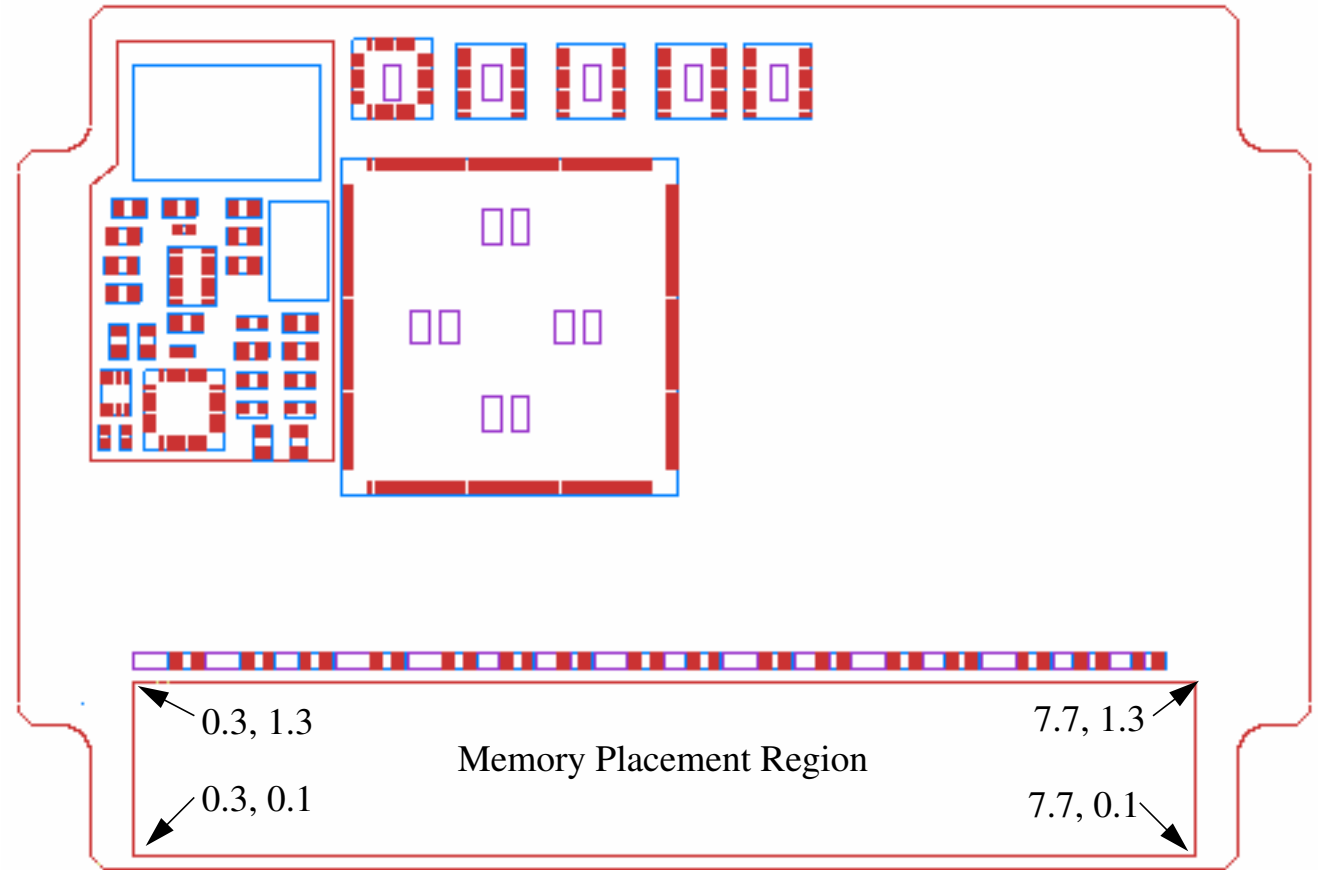


Figure 4-13. Board Placement Regions

Recall that placement rule areas are geometry attributes. If you need to change the placement region after you create it, you first set the select filter to select Geom Attributes. Then, select the placement area and delete it. Finally, recreate the placement area.

Autoplace the Memory Region

1. Choose the **[Top Menu] Auto Placement > Place Selected Components...** menu item. In the dialog box, enter the following and **OK** the dialog box.

   Place on: **Top Only**
   Place Components: **In Minimum Area**
   Orientations: **Try All Legal Orientations**
   Select only **180** as a legal orientation.

   The autoplacer cannot place all the memory components within the placement rule area without causing clearance violations. In the following steps, you make adjustments to the placement.

2. Unselect the components.

Adjust the Placement

You want the AM2168 components to be in groups as shown in Figure 4-14. There is approximately.1 spacing between the groups.

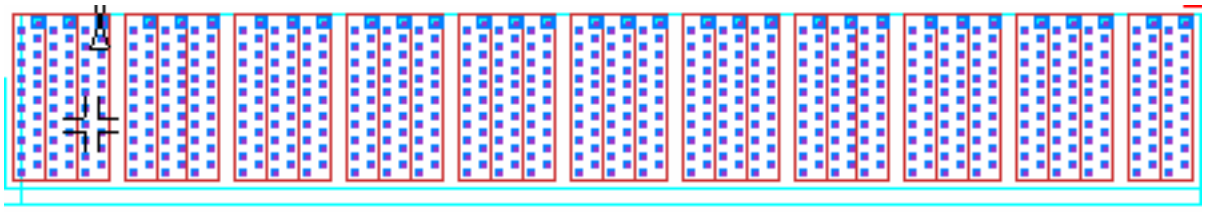
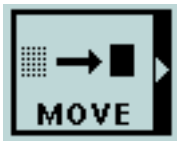


Figure 4-14. RAM Components on Board

Moving the components up against one another like this violates the placement clearance even though this is the best placement for the components. You achieve this placement by turning checking off, adjusting the components, then overriding placement violations for these components.

1. Choose menu item **Setup > Automatic Checking Off**.



2. From the AutoPlace palette menu, select the **Move** icon. From the choices in the Move menu, select **Move From To**.

   Now, adjust the placement of the components.

3. The quickest way to group the components is to take every three components on the board and butt them up against one another. Also, group the three unplaced components just below the board outline. You end up with one group of two, as there are 32 AM2168 components.

4. When you have the components in groups, start at the right side of the board, select and move each group to the right, leaving approximately .1 spacing between each group. Your placement does not need to look exactly like the placement in Figure 4-14.

5. When you get to the left side of the board, you should have enough space to move the group of three unplaced components onto the board. It is not necessary that the three unplaced components fit precisely within the placement region. The placement region is only to facilitate autoplacement.

6. Click on a component to select the component.

7. Click on the component again to indicate where you are moving it from, then move it to the desired location. Click at the new location.

8. Repeat this process for the other components.

9. When you finish, **Cancel** the prompt bar.

Align the Memory Components

To facilitate routing, align the memory components.

1. From the AutoPlace palette menu, choose **Align**.

   The Select Area prompt bar displays.

2. Select all of the AM2168 components.

3. Fill out the Align Components dialog box as follows.

   Align Components in a: **Row**
   By: **Top Most** By: **Pins**
   Align to: **Left Most Component**
   Repeat: **[unselect the button]**

4. **OK** the dialog box.

Ignoring Clearance Violations

If you perform a Check Components at this point, you definitely get clearance violations on the AM2168 components you placed up against one another. But, you know this placement is fine. You want to tell LAYOUT to ignore any clearance violations for these components.

First, check the placed components without ignoring placement violations for the components in the memory section.

1. Choose **Check > Components > Check Components**.
2. Select **Placed Components Only** and **OK** the dialog box.

   You receive errors for the components in the memory section. The Violations Ignored section contains the reference designators of the resistors placed on the right side of the board. LAYOUT is ignoring placement violations for the resistors, as the designer specifically placed them up against one another. You are going to do the same for the AM2168 components.
3. Select all of the AM2168 components.
4. Choose **Check > Components > Ignore Placement Violations**.

   When you check the components again, you receive no errors.
5. Choose **Check > Components > Check Components**.
6. Select **Placed Components Only** and **OK** the dialog box.

   The reference designators for the AM2168 components are listed in the Violations Ignored section of the report along with the resistors.
7. Unselect the components and turn automatic checking back on.

Defining Placement Sites

In the next few steps, you select and place four components. Before placing the components, you define a grid pattern on which to place the components and a boundary within which to place them.

1. Choose the **Setup Placement > Auto Placement > Define Placement Sites:** menu item. In the prompt bar, enter the following:

   X Grid: **0.7** Y Grid: **0.7**

   TAB to the Origin prompt.

2. Enter an origin by placing the cursor at absolute coordinates **1.7, 2.1** and clicking the Select mouse button.

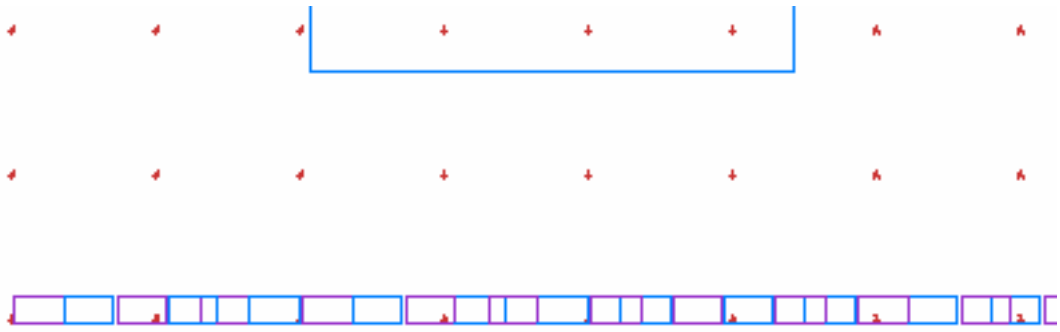


Figure 4-15. Placement Sites

The origin you enter snaps to the closest placement grid and becomes the first placement site.

Creating a Placement Boundary

Placement boundaries are temporary. Placement rule areas are permanent.

You can create a temporary placement boundary to facilitate automatic placement.

1. Choose menu item **Setup Placement > Auto Placement > Auto Placement Boundary.**
2. Using the Select mouse button, draw a rectangle with an upper-left coordinate of **1.2, 2.5** and a lower-right coordinate of **4.3, 1.6**.

The temporary boundary displays surrounding four placement sites.

Select and Place the Components

1. Use the **[Auto Placement] Select > Select By Name...** menu item to select the components with the **Ref** property with values of:

   U64

   U65

   U66

   U67

   You have 4 components selected even though you have a select count of 8. Each component and its reference designator are selected separately resulting in the select count of 8.

2. Choose the **[Auto Placement] Place Selected Components...** menu item. In the dialog box, enter the following and **OK** the dialog box.

   Place on: **Top Only**
   Place Components: **In Minimum Area**
   Orientations: **Use Smallest Orientation**

   The ROM components are placed in the placement boundary with their origins on the placement sites.

3. Unselect the components.

Clearing Placement Boundary and Sites

Now delete the placement boundary. If a placement boundary exists, the autoplacer sees the area within the boundary as the entire board and does not place any components outside the boundary.

1. Choose menu item **Setup Placement > Auto Placement > Clear Auto Placement Boundary**.

   Also, clear the placement sites.

2. Choose menu item **Setup Placement > Auto Placement > Clear Placement Sites**.

Adding the Dec_cap Property

For the autoplacer to place decoupling capacitors near chip components, the components need to have the Dec_cap property attached. In this section, you map the remaining components to an area outside the board outline, then attach the Dec_cap property to the chip components.

1. Choose **Map Components** from the AutoPlace palette menu.
2. Fill out the Map Components dialog box as follows, then select **OK**.

   Top Only
   Map Components to Area: **Outside Board**

*To find out which components you mapped to? Select the components and choose the menu item **Report > Selected**.*

3. Select the chip components in the mapped area.
4. Choose menu item **Properties > Component Properties > Dec_cap**.
5. In the Add Dec_cap Property dialog box, enter a Dec_cap part number of **BHR-1X0000-23**.
6. **OK** the dialog box.
7. Unselect the components.

Place the Remaining Components

Place the remaining components on the board using any of the placement techniques you wish. Experiment with both automatic and interactive placement. Here are some ideas.

- Create a placement boundary, select some components, and autoplace the selected components.
- Place the decoupling capacitors on the back of the board. The Place Components and the Place Selected Components dialog boxes each have a Place On section where you can specify the side of the board on which to place components.

And remember that you can turn Automatic Checking off temporarily to more easily make placement adjustments. Turn checking back on to ensure you have not created placement violations.

Adding Discretes

You can add decoupling caps as necessary and back annotate them to the schematic.

LAYOUT provides the capability of adding decoupling capacitors to the design when a layout requires more than are currently provided for in the design. Recall the constraints for adding decoupling caps.

- The loaded catalogs must contain the part number you wish to add.
- The *geoms* design object must contain the geometry for the component.
- The pins of the added decoupling capacitor must be connected to global nets, such as a power or a ground net.
- A global symbol with a name that exactly matches the global net name must exist in the symbol search path.
- Both the symbol name and the symbol's Comp property must match the Comp property of the discrete component on the board.

In this section, you add eight decoupling capacitors to your design.

1. In the Edit Window, choose menu item **[Top Menu] Placement > Extended Menu > Add Discrete Components**.

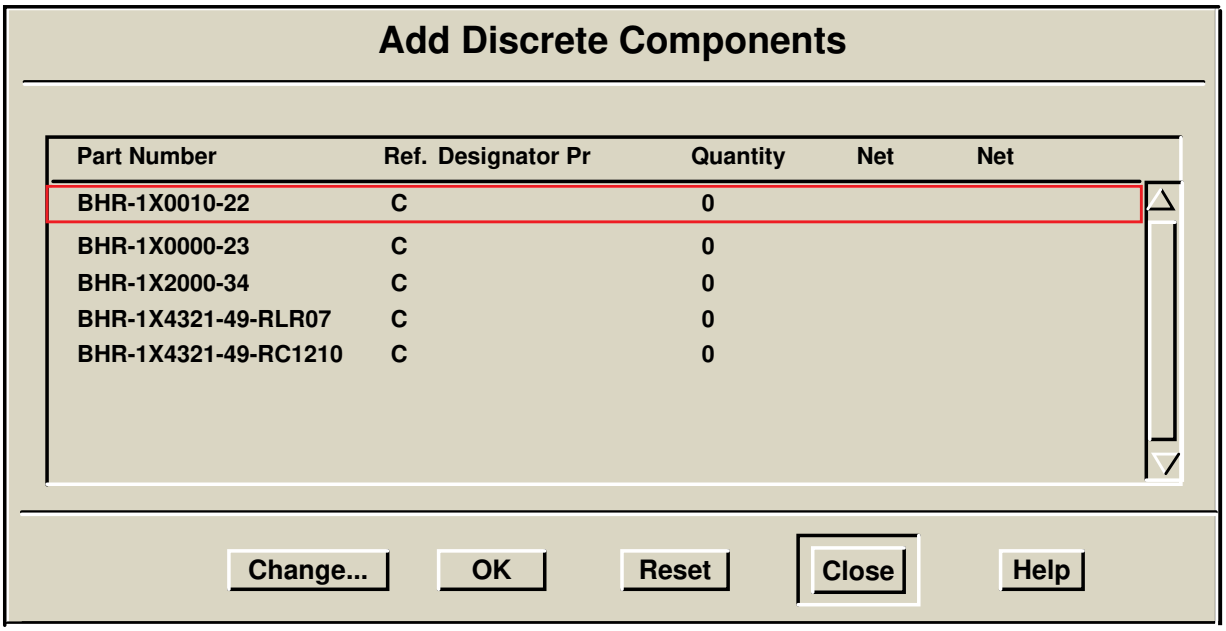


Figure 4-16. Add Discrete Components Dialog Box

The Add Discrete Components dialog box displays. All the discretes in the loaded catalogs display in the dialog box.

2. From the list box, select part number **BHR-1X0000-23**.

3. Select to **Change...**

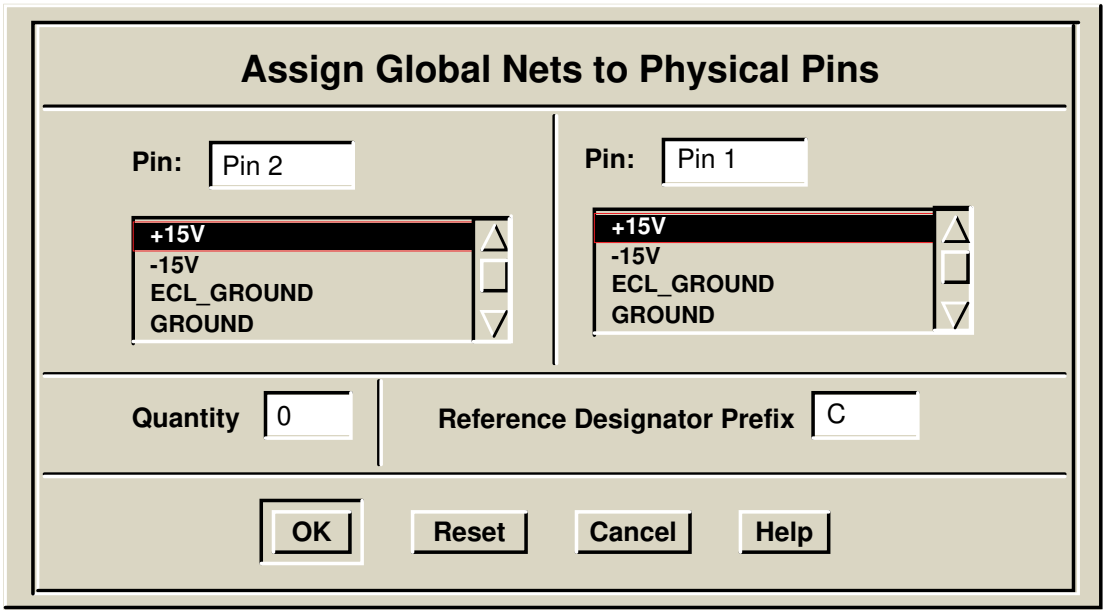


Figure 4-17. Assigning Global Nets to Physical Pins

The Assign Global Nets to Physical Pins dialog box displays. This is where you specify the number of components of this part number to add to the design. You also specify the net to which to attach each pin.

4. Assign Pin 2 to **Ground** and Pin 1 to **VCC**.

5. In the Quantity box, add **8** components to the design and use the default reference designator prefix of **C**.

6. **OK** the dialog box.

   Note that the Add Discrete Components dialog box now updates showing that eight components of part number BHR-1X0000-23 are being added to the design.

7. **OK** the Add Discrete Components dialog box.

OKing the dialog box adds the discretes to the design. Closing the dialog box instead does not add the discretes to the design.

Place the Discretes

1. Map the discrete components to an area outside the board outline on the back side of the board.

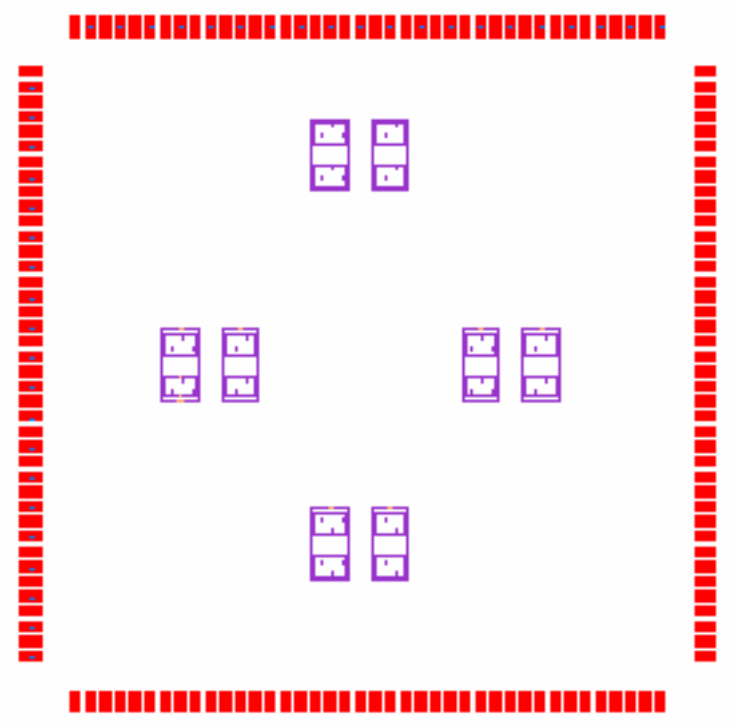


Figure 4-18. Discrete Placement on Back Side of Board

2. Move the discretes into the configuration shown in Figure 4-18.

If you placed the decoupling capacitors on the back of the board, you might need to move some components to place these discretes in the configuration shown in Figure 4-18.

Changing a Geometry

You can change the geometry or part number used by a symbol. Remember that to change a geometry or part number:

- The geometry must be in the *geoms* design object.
- The part number must be in a loaded catalog file.

New geoms and part numbers must already be part of the design.

In this case, you change a geometry from a surface mount geometry to a through-pin geometry.

1. Use the **[Top Menu] Auto Placement > Select > Select By Name...** menu item to select the components with **Comp** property **74AS245**.
2. Make a note of which components highlight, then Unselect All.
3. Select one of the components with Comp property 74AS245.

4. Select **[Top Menu] Placement > Extended Menu > Change Geometry**.

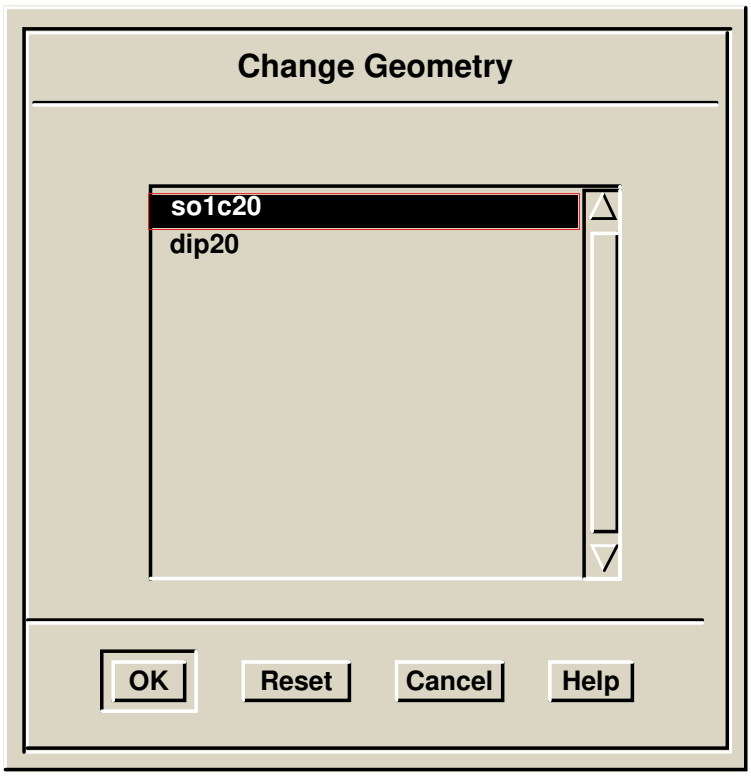


Figure 4-19. Changing a Geometry

5. In the list box that appears, select the **dip20** geometry and **OK** the dialog box.

The component now uses the dip20 geometry. You can judge which geometry works best for the layout. You can either keep the dip20 or change the geometry back to the soic20 geometry.

Changing the geometry can unplace the component. If this happens, decide which geometry to use and place the component back on the board.

Reducing Trace Length

You can now use the Auto Swap feature to evaluate the trace lengths and perform swaps to reduce the trace length.

1. Choose the **[Auto Placement] Auto Swap > All Components...** menu item. In the dialog box, choose **Swap Unlike Geometries** and **OK** the dialog box.

You can swap components, gates, pins, or pinsets.

Swap takes a few moments, but you can monitor the number of exchanges in the Status window. When the swapping is complete, the net length reduction and new Manhattan length display in the Message window at the bottom of the Session window.

Components in the memory region and along the side of the board now highlight in the error color. Because geometries were swapped, these components now cause placement violations. You need to tell LAYOUT to ignore these violations as you did earlier.

2. Select all of the components causing violations.
3. Choose **Check > Components > Ignore Placement Violations**.

   When you check the components again, you receive no errors.

4. Choose **Check > Components > Check Components**.

   Now try another auto swapping feature: swapping all gates. Autoswapping uses the gate swap code you specified in the mapping file.

5. Choose the **[Auto Placement] Auto Swap > All Gates:** menu item. In the prompt bar, enter **3** as the Number of Passes and select **OK**.

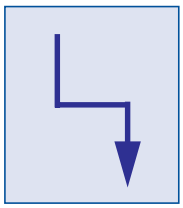
   This swap is done quickly. The net length reduction is displayed in the message window.

   Now swap swappable pins. This uses the pin swap code specified when creating the geometry.

6. Choose the **[Auto Placement] Auto Swap > All Pins** menu item.

Displaying Density

The Density object provides visual feedback on the density of the currently visible objects on the board. The greater the density of the viewed objects in a specific area of the board, the brighter the shading in that area.



1. View the entire board in the Edit window.

2. Select **View > Display Controls** and select to view the **Density** object.

3. Now select **Set Display Attributes**. For Density, set the pattern to solid and select a bright color, such as a bright red. **OK** the Set Display Attributes dialog box. Also **OK** the Display Controls dialog box.

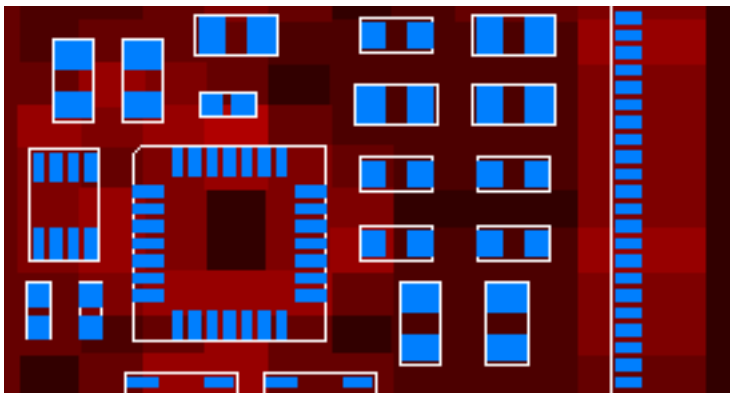


Figure 4-20. Shades of Density

Note the shading differences. The brighter shades reflect areas of greater density. Recall that density reflects the density of only the viewed objects on the board.

4. Use the Display Controls dialog box to view or not view various objects. See how what you select to view or to not view affect the density shading on the board.

While the Density object provides a visual reference to identify more and less dense areas of the board, you can also generate a histogram and a routability report to evaluate the placement and routability of your board.

Evaluating Placement

Next, you create and view a histogram to evaluate component placement.

1. Choose the **Report > Placement Histogram** menu item.

   Histogram skylines showing board congestion display. If histogram bars exceed 40%, you should adjust the placement in that area of your board design.

2. Choose the **Report > Placement Histogram > Clear Report** menu item to clear the histogram from the display.

3. Choose the **Report > Routability** menu item. **OK** the dialog box with the default information.

   When the report is complete, the report window displays.

Routing evaluation allows you to re-place congested areas before routing.

This function takes a few minutes to process the board placement data. You can monitor the progress by watching the status window. The result of this function is a routing density display. The density display indicates the location of areas that can be difficult to route. A density of 100% indicates that all of the available routing channels are filled. Any 100% or 90% areas indicated on the board can be more difficult to route than other areas. If you have areas indicated to be 100% or 90% filled, you should consider moving components to provide more space for routing.

4. Close the report window. Use the View All stroke to see the scale range on the left side of the board.

5. After looking over the scale range, clear the report by choosing the **Report > Routability > Clear Report** menu item.

You generate another routability report after routing breakouts in the next module.

Back Annotation Including Discretes

Along with other changes, you need to back annotate to add the new discretes to the schematic. First, you need to set up parameters for annotating discrete components to a schematic sheet.

1. Choose menu item **Setup > Schematic > Discrete Annotation**.

   The Setup Discrete Annotation dialog box appears.

2. In the Schematic Name entry box, type **schematic**.

   This is the container where the schematic sheet is located under the design container. Schematic is the default, so it might already be in the entry box.

You define the sheet for the discretes.

   In the Sheet Name entry box, type **sheet_dec_cap** as the name of the new schematic sheet for the discrete gates.

   This sheet is created under the schematic container that you specified in the previous step.

3. Specify a Sheet Width of **8.5** and a Sheet Height of **11**. Select **Inches** as the units of the sheet.

4. Specify all margins as **0.5**. Enter an x-increment and a y-increment of **0.25**.

5. Choose **Specify** to set a Symbol Search Path.

   In the Path typing box, enter the directory name: **parts**

   In the next Path typing box, enter the directory name: **gen_lib**

6. **OK** the dialog box.

7. Choose menu item **File > Back Annotate**.

8. Set the Back Annotate dialog box as follows and select **OK**.

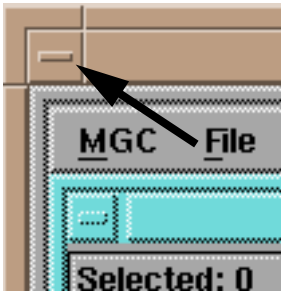
   Back Annotate Discretes: **Automatic**.

   Back annotating creates the sheet with the discrete gates.

9. Choose menu item **File > Save > Design Specify** and select to save the **Environment**. This saves the settings established in the Setup Discrete Annotation dialog box.

10. Choose menu item **File > Save > Design All**.

Closing LAYOUT



1. Close the session by choosing **Close** from the Window menu (upper-left most window icon).

   The **Save Changes to Design** dialog box appears. Because you have already saved your data and back annotated to the PCB design viewpoint, it is not necessary to save the data again. However, it is good design practice to save data and back annotate as you leave LAYOUT to ensure the layout and the PCB design viewpoint remain in sync.

2. Choose **Yes** and the dialog box progresses to ask if you want to **Back Annotate** the **PCB Design Viewpoint**.

3. Choose **Yes** and the dialog box allows you to **Back Annotate Discretes** in one of two ways.

   - **Automatic** back annotates discrete components only if discrete components were added to the design and not previously back annotated.

   - **Manual** back annotates discretes whether the discretes have already been back annotated or not.

4. Choose **Automatic** and **OK** the dialog box.

Congratulations! You have completed the *Placing Components on a Circuit Board* module. The next module of the *Board Station for New Users Training Series* is Module 6: *Routing Traces on a Circuit Board*.

Appendix A: PCB Layer Definitions

When you invoke LIBRARIAN, LAYOUT, or FabLink on a design, the tool reads layer definitions from the *layers* design object in the *pcb* container of the design. If the design object does not exist or if you invoke LIBRARIAN without a design, the tool reads the *layer_file* in your working directory. If neither the *layers* design object or the *layer_file* exist, the tool reads the default layer file from *$MGC_HOME/pkgs/pcb_base/data/layer_file*.

You can modify a layers design object or a user-specific layer_file to suit your needs. Your System Administrator can modify the *$MGC_HOME/pkgs/pcb_base/data/layer_file* to suit the needs of your site.

Layer Data Parameters

Table A-1. shows a portion of the default *layer_file* found in *$MGC_HOME/pkgs/pcb_base/data/layer_file*. The *layer_file* is in an ASCII file containing the name, stacking number, and several display parameters for each layer in Board Station. Each layer requires a value for each parameter listed.

Table A-1. Layer Data Parameters

Name	Stack #	Type	Color	Width	Fill	Patt	Trans	Path	Space
Board_outline	10	user	1	1	26	2	0	0	0
Breakout	16	user	7	1	12	12	0	0	0
Breakout_1	39	user	8	1	12	1	0	0	0
Breakout_2	40	user	9	1	12	1	0	0	0

Parameter Definitions

- Layer Name—a unique name that identifies the layer in the file. This name displays in the various layer dialog boxes.
- Stacking Number—a unique number in the layer stacking order.
- Type—the only type currently supported is User.
- Color—the color index of the graphic object displayed on the layer. Taken from the Color Map Table.
- Width—currently not implemented.
- Fill—currently not implemented.
- Pattern—the fill pattern of primitive graphic on the layer. This value is a number that corresponds to one of the patterns in the Change Layer Display Attributes dialog box.
- Transparency—sets the transparency of the fill pattern. Transparent fill patterns mean that objects behind a filled object can be seen through the filled object.
- Path—currently not implemented.
- Space—currently not implemented.
- Style—the line drawing style used on the layer. 1 indicates a solid line. 2 indicates a dotted line. 3 indicates a short dash. 4 indicates a long dash.
- Text—currently not implemented.
- Pen—associates a plotter pen with the layer.
- Hilite—the color of all highlighted objects on the layer.
- Protect—the color of all protected objects on the layer.
- Select—the color of all selected objects on the layer.

Layer Names and Numbers

It is important to know which stacking numbers are already used by Board Station layers. You can define a new layer, but you must assign an unused stacking number. **Table A-2.** shows the currently used Board Station layers and the associated stacking numbers.

Table A-2. PCB Layers and Numbers

Layer Name	Stacking #
Board_outline	10
Breakout	16
Breakout_1	39
Breakout_2	40
Dam	138
Dam_1	139
Dam_2	140
Density	120
Density_1	121
Density_2	122
Drawing	8
Drawing_1	33
Drawing_2	34
Drill	5
Drill_holes	18
Errors	17
Force	237
Glue_mask	201
Glue_mask_1	202

Table A-2. PCB Layers and Numbers [continued]

Layer Name	Stacking #
Glue_mask_2	203
Guide	50
Histogram	238
Milling	242
Off_grid_pins	135
Off_grid_pins_1	136
Off_grid_pins_2	137
Pad	46
Pad_1	47
Pad_2	48
Pad_master	4
Panel_outline	21
Paste_mask	12
Paste_mask_1	37
Paste_mask_2	38
Pin_id	11
Pin_id_1	28
Pin_id_2	29
Place	1
Placement_keepout	15
Placement_region_1	141
Placement_region_2	142
Place_1	31

Table A-2. PCB Layers and Numbers [continued]

Layer Name	Stacking #
Place_2	32
Power	3
Power_1	81
Power_2	82
Power_master	7
Route_grid	150
Route_grid_1	151
Route_grid_2	152
Routing_keepout	14
Signal	2
Signal_1	51
Signal_2	52
Silkscreen	6
Silkscreen_1	41
Silkscreen_2	42
Solder_mask	9
Solder_mask_1	35
Solder_mask_2	36
Thermal	130
Thermal_1	131
Thermal_2	132
Trace_keepout	20
Via	13

Table A-2. PCB Layers and Numbers [continued]

Layer Name	Stacking #
Via_1	91
Via_2	92
Via_keepout	19
Via_usage	90

User-specific Layers

You can modify a layers design object or a user-specific layer_file to suit your needs. In addition, your System Administrator can modify the *$MGC_HOME/pkgs/pcb_base/data/layer_file* to suit the needs of your site.

For example, you design requires more signal layers than the 12 signal and 8 power layers provided in the default layer_file. You can add layers to the *layers* design object for that design.

You can modify the *layers* design object in three ways:

- Add layers using the **Setup Design Rules > Logical Layers** menu selection in the LIBRARIAN tool.
- Edit the layers design object in any PCB tool with the PCB Data Editor using the **View > Design Data** menu selection.
- Edit the *layers* design object with the operating system text editor. You can also use this procedure to modify your *layer_file* when not invoked on a design.

Use the following guidelines when modifying the *layers* design object or the *layers_file*.

- Only add new layers. Deleting unused layers is not necessary.
- Use new layer names for the layers you create. A new layer name must not exceed 40 characters. The first character of the layer name must be one of the following characters: a-z, A-Z, 0-9, or $.
- Use consecutive stacking numbers for layer sets. If the new layer is an addition to a layer set, assign the next consecutive number. For example, Signal_12 uses a stacking number of 62. When adding another signal layer called Signal_13, assign the number 63.
- When adding a stand-alone layer, rather than adding to a layer set such as the Signal_n layer set, use a stacking number that is not already assigned to another layer and is above 200 but below 255. Also do not assign stacking numbers 201, 202, 203, 237, 238, 241, and 242.
- When using a text editor to add a new layer, be sure to define each parameter shown in the Layer Data Parameters section of this appendix.

Appendix B: Design Object Data Organization

Appendix B discusses design data organization. PCB design objects organize the design data for a specific design. Each design object represents a group of related design data, such as files and directories. Design objects that represent directories are called containers. **Figure B-1.** shows the organization of data for a PCB design.

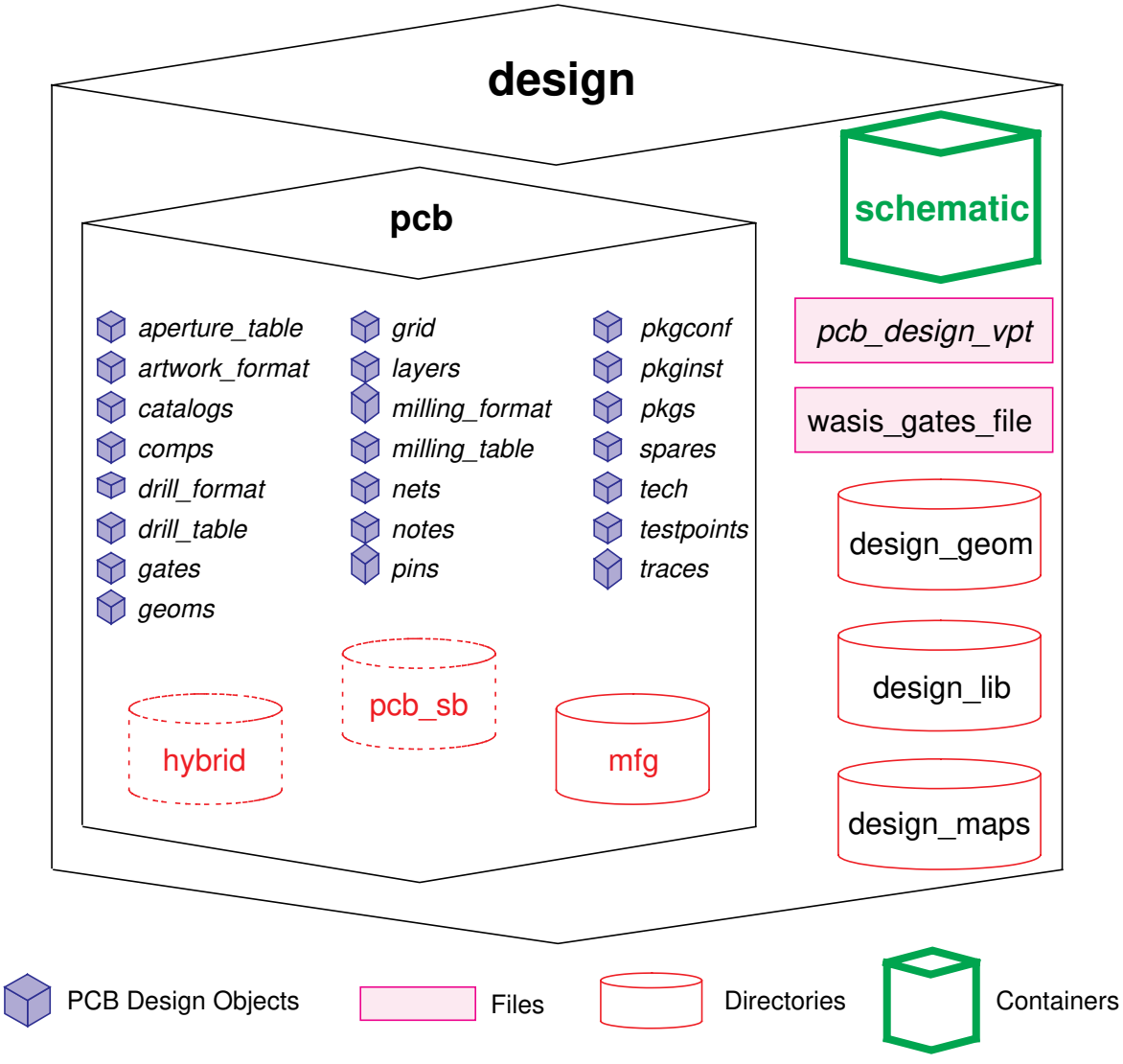


Figure B-1. PCB Design Data Organization

Design Object Versioning

The PCB tools create and maintain multiple versions of design objects. This allows you to recover data from a previous version of a design object, if required. Each time a PCB tool saves or updates the data in a design object, the system creates a new version of the design object. **Figure B-2.** shows versioning of the comps design object. Each new *comps* design object version creates a new *comps.comps*_n file. The _n suffix on the filename indicates the design object's version number.

The system maintains three versions of PCB design objects by default. This is called version depth. The system deletes older versions of the design objects as the PCB tools create new versions. You or your system administrator can change the version depth. Refer to the *Design Manager User's Manual* for instructions. You can also freeze a version of a design object to prevent the deletion of that version.

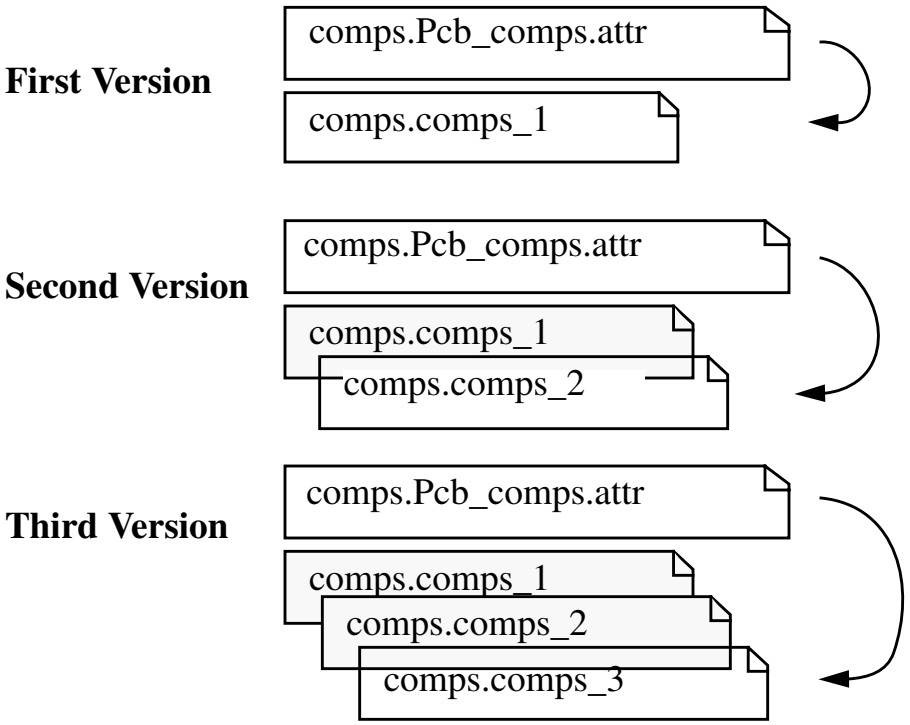


Figure B-2. Comps Design Object Versioning

The comps.PCB_comps.attr file always references the most recent version of the comps.comps_n file.

Design Object Data

The design data specific to a PCB design reside in the related design object. This includes the schematic data as well as the PCB design data. This section describes the data for a PCB design. Refer to the *Design Architect Reference Manual* for more information on the schematic design data.

Table B-1. contains an alphabetized summary of the PCB design data in the design objects in the *pcb* container. This table does not include manufacturing output data generated by FabLink. Refer to the *Fablink User's Manual* for descriptions and examples of the manufacturing output that FabLink generates.

Table B-1. Design Object Data Summary

File	Purpose	Filename	Created By:	Used By:
Aperture table data	Establishes aperture settings on an aperture wheel	*aperture_table*	FabLink	FabLink
Artwork format data	Describes how the artwork data is written to artwork files	*artwork_format*	FabLink	FabLink
Catalogs data	Contains names of catalogs to load when PACKAGE invokes	*catalogs*	PACKAGE User	LAYOUT FabLink
Components data	Lists the components needed for the design	*comps*	PACKAGE	PACKAGE FabLink
Drill format data	Describes how drill data is written to files	*drill_format*	FabLink	FabLink
Drill table data	Maps drill bits to the positions in a drill magazine	*drill_table*	FabLink	FabLink
Gates data	Lists the mapping and swapping information for all symbols in the design	*gates*	PACKAGE	LAYOUT FabLink
Geometries data	Describes all geometries needed for the design	*geoms*	PACKAGE LIBRARIAN	LAYOUT FabLink
Grid data	Describes the routing grid from a previous LAYOUT session	*grid*	LAYOUT	LAYOUT

Table B-1. Design Object Data Summary [continued]

File	Purpose	Filename	Created By:	Used By:
Layers data	Specifies the arrangement and characteristics of data layers	*layers*	LIBRARIAN LAYOUT FabLink	LIBRARIAN LAYOUT FabLink
Milling format data	Describes how milling data is written to files	*mill_format*	FabLink	FabLink
Milling table data	Establishes the mapping of milling tool sizes to tool positions in a milling machine	*mill_table*	FabLink	FabLink
Nets data	Lists the component pin connections for the design	*nets*	PACKAGE	User
Notes data	Lists all pin, gate, and net notes in the schematic	*notes*	PACKAGE	User
Package configuration data	Controls how PACKAGE treats properties and their values from the schematic	*pkgconf*	LIBRARIAN User	PACKAGE LAYOUT FabLink
Package instance data	Copy of the gates data used for creating the was-is file	*pkginst*	PACKAGE	LAYOUT
Package data	Contains a summary of the packaging process	*pkgs*	PACKAGE	User
Pins data	Lists pin properties	*pins*	PACKAGE LAYOUT	LAYOUT
Spares data	Lists spare and reserve parts	*spares*	PACKAGE	User
Technology data	Correlates physical layers with data layers; defines rules for nets, vias, and pins	*tech*	LIBRARIAN LAYOUT FabLink	LIBRARIAN LAYOUT FabLink
Traces data	Contains the routing and area fill information for the design	traces	LAYOUT	LAYOUT FabLink

Appendix C: PCB Products System Limitations

Appendix C lists the constraints and tested limitations of the PCB tools design software.

Table C-1. Component Limitations

COMPONENTS	Maximum
Total pins on a component	3700
Power pins on a component (POWER statements in mapping file)	500
Non-power pins on a component (PIN statements in mapping file)	3200
Portions in a component (SYMBOL statements in mapping file, including edge connectors)	2000
Different symbols in a non-homogeneous part (Comp properties in mapping file)	20
Gate swap code values in a mapping file	0 - 4095
Pins in a pin set	50
Check pins for a pin set	50
Pins sets for a part	50
Aliased power nets	200

Table C-2. Design Limitations

DESIGNS	Maximum
Total part numbers in a design	5000
Potential part numbers for a symbol (for example, a resistor)	5000
Different Comp properties (symbols) in a design	2000
Instances (including spares) in a design	40,000
Sheets in a design	500
Total number of swappable part numbers and number of packages having the Swapping property (with the local value) in a design	4095
Placement regions in a design	200
Different Power_nets property values in a design	255

Table C-3. Property Limitations

PROPERTIES	Maximum
Different properties in a design	4000
Different properties for gates and components	2000
Different properties for nets	1000
Different properties for pins	1000

Table C-4. Property Limitations

CATALOGS	Maximum
Total number of catalog files in a PACKAGE session	1000
Search directories in a PACKAGE session	50

Appendix D: PCB Layout Properties

Appendix D lists properties conveying information meaningful to the Mentor Graphics PCB Layout tools.

Table D-1. PCB Layout Properties

PROPERTY	Owner	Description
Brd_loc	Instance (DA); Component (PCB)	Specifies a component's location on a printed circuit board
Comp	Instance (DA); Component (PCB)	Identifies a schematic symbol
Conn_order	Net	Explicitly orders a pin on a net
Dec_cap	Instance (DA); Component (PCB)	Associates the part number of a decoupling capacitor with a component
Elec_class	Net	Associates a net with an electrical class
$G	Instance, Component, Net, or trace	Generated by the system when you group or protect objects in LAYOUT or FabLink
Geom	Instance	Determines the geometry of the component to which the instance is assigned
Ink_id	Instance of a resistor	Hybrid property used by the resistor generator; the Ink_id property indicates the name of the ink used for the resistor
Inst	Instance	Unique name for an instance of a schematic symbol
Instpar	Instance of an analog component	Specifies analog device attributes in SPICE netlist format
Laser	Instance of a resistor	Hybrid property used by the resistor generator; the Laser property represents the type of laser trim used in the fabrication process
Match_group	Net	Associates a net with a match group

Table D-1. PCB Layout Properties [continued]

PROPERTY	Owner	Description
Max_stub	Pin	Overrides the maximum stub limitations set at the electrical class or group level, for a specific pin
Min_stub	Pin	Overrides the minimum stub limitations set at the electrical class or group level, for a specific pin
$MP	Instance (DA); Component (PCB)	Used by PCB tools to keep track of the new location of a moved pin
Net	Net	Specifies signal connectivity by assigning unique names to wires, buses, ports, and offpage connectors
Net_length	Net	Specifies high and low limits on the length of a net
Net_order	Net	Specifies how the net should be broken up
Net_prio	Net	Specifies how critical is a specific net
Net_type	Net	Defines the name of a specific net_type
$NP	Instance (DA); Component (PCB)	Keeps track of a changed padstack on a component
Part_no	Instance (DA); Component (PCB)	Specifies the part number into which a symbol is packaged
Pcb_inst	Instance	Specifies a design note for a instance
Pcb_net	Net	Specifies a design note for a net
Pcb_pin	Pin	Specifies a design note for a pin
Pin	Pin	Name for a logical pin on a schematic symbol
Pin_group	Pin	Associates pins with a pin group
Pin_no	Pin	Specifies the physical pin number to which the logical symbol pin is assigned
Pin_order	Pin	Specifies the order of pins on a net
Pin_swap	Pin	Becomes the swap code for the pin in the mapping file
Pintype	Pin	Specifies the direction of signal a pin conducts
Placement_region	Instance (DA); Component (PCB)	Associates schematic symbols with a predefined region of the board for placement

Table D-1. PCB Layout Properties [continued]

PROPERTY	Owner	Description
$Pn_ext	Instance (DA); Component (PCB)	Keeps track of a changed geometry
Power_nets	Instance	Defines the name(s) of power net(s) on a component
Power_pins	Instance	Designates which physical pins connect to which power nets
Pow_max	Instance	Specifies the maximum amount of power the component is designed to dissipate
Pow_min	Instance	Specifies the minimum amount of power the component is designed to dissipate
Pow_typ	Instance	Specifies the typical amount of power the component is designed to dissipate
Ref	Instance	Specifies the reference designator
Refloc	Instance (DA); Component (PCB)	Identifies the changed location of a reference designator
Restrict	Net	Defines T-junction restrictions for a net
R_hat_len	Instance of a resistor	Indicates the hat length of a tophat resistor
R_hat_width	Instance of a resistor	Indicates the hat width of a tophat resistor
R_len	Instance of a resistor	Indicates the length of a resistor
R_min_dim	Instance of a resistor	Specifies the minimum dimension for a resistor
R_shape	Instance of a resistor	Indicates the shape to create the resistor
R_width	Instance of a resistor	Indicates the width of a resistor
Shared	Instance	Specifies that a symbol is shared by multiple symbol activations
Source	Net	Specifies the source pins of potential ECL nets
Surface	Instance of a resistor	Indicates the side of the substrate on which the resistor is placed
Swapping	Instance	Controls the gate and pin swap codes by overriding the swap codes in the mapping file
Tech	Instance	Identifies an instance as being ECL technology

Table D-1. PCB Layout Properties [continued]

PROPERTY	Owner	Description
Terminator	Net	Indentifies a terminating resistor pin for an ECL net
Toler	Instance of a resistor	Indicates the resistor value tolerance
Trim	Instance of a resistor	Used by the hybrid resistor generator to calculate the number of squares required for the resistor value
Value	Instance of a resistor	Used by the hybrid resistor generator; indicates the value of a resistor in ohms.

INDEX

A

P

R

S

欢 迎 光 临

http://www.mweda.com　　http://edastudy.ik8.com

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A　ADS2003C　ADS2003A
HFSS9.2　HFSS9.1　HFss9.0　HFSS8.0
Ansoft Designer1.1　Ansoft Serenade8.71　Ansoft Maxwell 10　Ansoft SIWave
Microwave Office2002　Sonnet Suite Pro 9.52　CST5.0
Super NEC 2.5　Zeland IE3D9.2　XFDTD 6.0

破解软件类：PCB 工具软件

Mentor EN2004　Mentor EN2002　Mentor ePD2004　Mentor SDD2004
Mentor WG2004　Mentor WG2002　Mentor ISD2004
PowerPCB5.0　PowerLogic5.0　Orcad10.3
PADS2005　PADS2004
Cadence SPB15.2 (Allegro 15.2)　Cadence PSD15.0 (Allegro 15.0)

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程
Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册
Mentor Expedition (WG) PCB Training Workbook
Mentor DxDesigner Design Processing Training Workbook

HFSS： HFSS 9.2 入门与提高教程
HFSS 9.0 入门与提高教程
HFSS9 视频教程
HFSS9 Training Tutorials
HFSS8 Training Manual
电子科大 HFSS8 教程

ADS: ADS 中文基础教程
ADS 设计实验教程
ADS 入门教程
ADS2003 Training Workbook: ADS2003 fundamental
ADS Training Workbook for Momentum
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

Ansoft Designer:

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

Cadence Allegro:

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注： 本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D，Zeland IE3D, PowerPCB，Orcad等的培训教程。以及射频/微波/天线类英文原版书籍。详情可登陆：http://www.mweda.com 或 http://edastudy.ik8.com